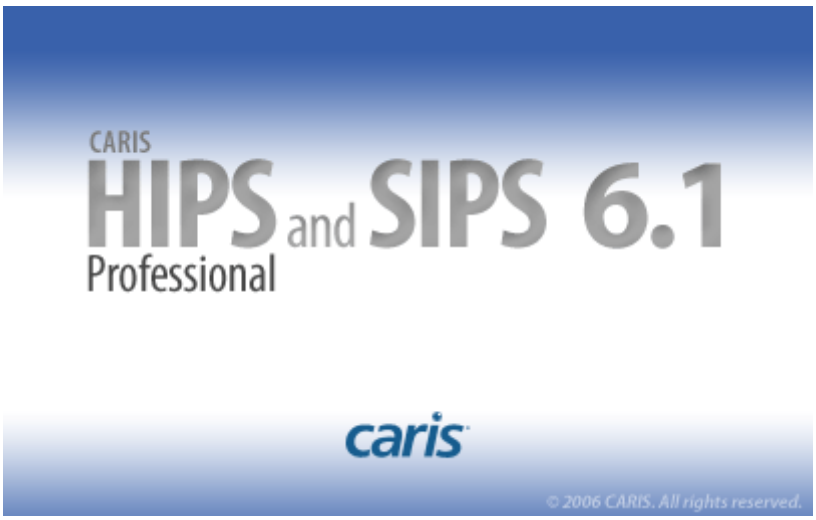


HIPS and SIPS 6.1 SP2
User Guide

Trademarks

CARIS software and the CARIS logo are registered trademarks of CARIS.

Other brands and product names are registered trademarks or trademarks of their respective holders.

Data courtesy of the Shallow Survey Conference Common Data Set 2001 (CCOM - UNH).

CARIS
(Global Headquarters)
115 Waggoners Lane
Fredericton, NB
Canada
E3B 2L4

Phone: 1 (506) 458-8533
(English/French/Spanish)
Fax: 1 (506) 459-3849
Email: info@caris.com
Assistance: support@caris.com
Web site: www.caris.com

CARIS bv
Mgr. van Oorschotstraat 13
PO Box 47 5473 ZG
Heeswijk, NETHERLANDS

Phone: +31 (0)413 296 010
Fax: +31 (0)413 296 012
Email: sales@caris.nl

CARIS USA
415 N. Alfred Street
Alexandria, VA
United States
22314

Phone: 1 (703) 299-9712
Fax: 1 (703) 299-9715
Email: carisusa@caris.com

CARIS Asia Pacific
Suite 1, Innovation House
Mawson Lakes Boulevard
Mawson Lakes SA 5095
Australia

Phone: +61.(0).8.8260.8180
Fax: +61.(0).8.8260.8100

Copyright 2008 CARIS

The material in this manual is for information purposes only and is subject to change without notice. CARIS assumes no responsibility for any errors that may appear in this manual. CARIS hereby grants licensees of CARIS software the right to reprint this document for internal use.

June 2008

Preface

Who should use this guide?

This user's guide describes how to operate the Hydrographic Information Processing System (HIPS) and the Sonar Information Processing System (SIPS) and should be read by all users.

Prerequisites

It is assumed you are familiar with hydrographic surveying concepts and the Windows NT/2000/XP operating systems. Some knowledge of digital hydrographic processing concepts is also assumed.

No prior experience of CARIS operations is required. However, HIPS and SIPS concepts are more easily grasped if you are familiar with other CARIS products.

What's in this guide?

The HIPS and SIPS User's Guide is divided into the following sections:

- **Introduction:** This section gives an overview of HIPS and SIPS.
- **HIPS Interface:** This section gives an overview of the interface, its windows and toolbars. It describes display options, how to select colours, modify toolbars and create custom keyboard commands.
- **Options:** This section describes the available options for modifying the interface, directories and environment variables.
- **Data Directories:** This section gives a brief description of the data directory structure in HIPS/SIPS.
- **HIPS Vessel File:** This section describes the various components in a vessel file. It also describes how to use the Vessel Editor to create new vessel files and edit existing files.
- **New Projects:** This section describes how to create a new Project-Vessel-Day directory structure for a HIPS/SIPS project.You will also learn about the various project options available in HIPS and SIPS.

- **File Conversion:** This section describes how to convert survey acquisition data into HIPS/SIPS format. The section explains how to use the File Conversion Wizard to convert data. It also describes the conversion options for each data format available in the wizard.
- **Survey Line Processing:** This section describes how to select track lines. The section gives an overview of the various editors and the commands for applying tide and sound velocity files. You will also learn about the Merge command for creating final position and depth files.
- **Navigation Editor:** This section describes the Navigation Editor and how to use it to clean the vessel's position data.
- **Attitude Editor:** This section describes how to use the Attitude Editor to clean and filter data from the vessel's motion sensors.
- **Sound Velocity Editor**: This section describes how to use the Sound Velocity Editor to revise Sound Velocity Profiles.
- **Swath Editor:** This section describes the various features found in the Swath Editor. It describes how to select and clean data. The section concludes with a description of the Refraction Editor.
- **Single Beam Editor**: This section describes the various features found in the Single Beam Editor. The section also includes a description of the barcheck function.
- **Tide Editor:** This section describes how to use the TidE Editor to revise tide files.
- **Field Sheets**: This section describes the function of the field sheet. It also describes the steps necessary to create one, and how to open and close field sheet data. The section also describes the options available for tile display.
- **BASE Surfaces**: This section describes the steps necessary to create a Bathymetry and Statistical Error (BASE) Surface. It also describes the property options of BASE Surfaces and the options available for editing BASE Surfaces.
- **Subset Editor**: This section describes how to create subsets and how to use the Subset Editor to view and clean sounding data.

- **CUBE Processing:** This section describes how to create a Combined Uncertainty and Bathymetry Estimator (CUBE) Surface. It also describes how to edit CUBE hypotheses data in the Subset Editor. The section concludes with a description of how to use CUBE filtering to reject outlier soundings.
- **Statistical Surface Cleaning**: This section describes how to divide selected areas into tiles and apply the statistical surface cleaning options to clean data. The section also describes the tiling and surface cleaning query options.
- **Lidar Data**: This section describes how to view and clean lidar data in both the Subset and Swath Editors.
- **Calibration**: This section describes how to use the calibration tool in HDCS (Hydrographic Data Cleaning System) to compare the effects of sensor offsets on calibration survey lines.
- **Side Scan Data**: This section describes the workflow needed to process side scan data in SIPS. The section also describes the Side Scan Editor and how to use this tool to create slant-range corrected data from raw side scan data. The section concludes with a description of mosaics.
- **Export Data**: This section describes how to export HIPS and SIPS data to other formats.
- **Batch Processor**: This section describes how to use the Batch Processing tool to automatically process data.
- **Appendices:** These provide reference material relevant to using the software.
- **Glossary**: This section contains an alphabetical listing of terms found in this guide.

Also see the CARIS Field Sheet Editor User's Guide for more information about processing field sheets.

Table of Contents

1 Introduction 1
Purpose of HIPS and SIPS 2
Development Partners 3
Design Principles 4
Licensing 5
HIPS I/O Library 6

2 HIPS and SIPS Interface 7
Opening HIPS and SIPS 8
Configuration Report 9
Main Display 10
Display Window 11
Control Window 12
Properties Window 14
Worksheet Window 15
Output Tab 15
Query Tab 16
Graph Tab 18
QC Reports Tab 21
Profile Tab 21
Reposition and Resize Windows 22
Status Bar 23
Status Bar options 23
Toolbars 24
Modify Existing Toolbars 25
Create a Custom Toolbar 26
Mouse Information 28
Open Projects and Background Data 29
Open a Project 29
Set Track Line Properties 30
Open Background Data 31
Set Image Properties 32
Display S-57 Data 34
Display Tools 37
Zoom 37
Zoom In 37
Zoom Out 38
Zoom/Pan Previous 38
Zoom/Pan Forward 38
Zoom To 39
Pan 39

Refresh Display 40
Overview Display 40
Show/Hide Layers 40
Selection Tools 42
Select by Range 42
Select All in Display 42
Select All 42
Select By Lasso 44
Select Next Line 45
Select Previous Line. 45
Select Lines From the Control Window 46
Clear All 46
Screen Captures 47
3-D Display Window 48
Set 3-D Window 49
3-D Display Properties 53
Control Axes. 58
Vertical Exaggeration 59
Navigate 3-D Display 60
3-D Display Fly-through options 63
Cut, Copy and Paste 66
Cut 66
Copy 67
Paste 68
Measure Distance 69
Keyboard Shortcuts 70
Save, Close and Exit. 72
Save Projects 72
Close a Project 72
Exit HIPS and SIPS 72

3 Options 73
Introduction 74
General 75
Display 78
Display window. 78
Geographic Grid 80
IHO Standards 82
Lidar 83
Projected Grid 84
Scale Bar 86
Sensor Editors 87
Side Scan Editor. 89
Single Beam Editor. 90
Status 92
Swath Editor 93
Tide/SVP 94

Units 95
Directories 97
Environment 99
S-57 Environment 101
S-52 Display 103

4 **Data Directories 107**
Introduction 108
Projects 109
HIPS and SIPS Data 109
VesselConfig Directory 110
Raw Data 111
SVP and Tide 112
SVP 112
Tide 112

5 **HIPS Vessel Editor 113**
Vessel Files 114
Working With the Vessel Editor 117
Open the Vessel Editor 117
Open, Close and Save Files 118
3-D Controls 119
Create Vessel Outline 121
Options Dialog Box 123
Exit the Vessel Editor 124
Create a New HVF 125
Vessel Information 125
Type of Survey 126
Single Beam 126
Multibeam 127
Multi-Transducer 128
Motion Sensors 129
Configuration Options 130
Sound Velocity Corrections 131
Edit Sensor Configuration 132
Swath 134
Navigation 135
Gyro 135
Heave 137
Pitch 138
Roll 139
Dynamic Draft 140
Single Beam 141
SVP 142
Sweep 143

Towed Sensors 145
Waterline 146
Total Propagated Error 147

6 **Starting a New Project 149**
Define Project 150
New Project (Step 1) 150
New Project (Step 2) 153
New Project (Step 3) 154
New Project (Step 4) 155
Rename Day and Line Folders 156
View Project or Vessel Properties 157
Session Files 159
Recycle Bin 161
Archive Project 163

7 **File Conversion 169**
Converting Files to HIPS/SIPS Format 170
Select Data Format 171
Select Files for Conversion 172
Select Project-Vessel-Day 173
Select Coordinate System 174
Select Basic Filters 175
Enter Data Parameters 176
Convert Data 177
Data Format Options 178
Atlas 179
Chirpscan3D 180
CMAX 180
Coda 180
EdgeTech 180
EIVA 181
Elac 181
Furuno 182
GeoAcoustics 183
GSF 185
Hypack 186
Imagenex 187
LADS 188
LAS 188
MarineSonics 189
Navitronics 189
Qmips 189
Reson PDS 189
Scripps 189

SDF 190
Seabeam 190
Seafalcon 190
SEGY 191
SHOALS 191
Simrad 193
SWATHPlus 194
UNB 194
Winfrog 194
XTF 195
Convert Bathymetric data 196
Convert Side Scan data 199

8 **Survey Line Processing 203**
Select Survey Lines 204
Query Line Data 205
Set Line Classification 206
View Rejected Soundings 207
Designated Soundings 208
Critical Soundings 210
Reject Line 211
Correct for Tide 212
Tide Files 213
Tide Zone Files 213
Load Tide 215
Attitude Sensor Data 217
Attitude Filter 217
TrueHeave Data 219
Load Attitude/Navigation Data 221
Sound Velocity Processing 223
SVP Files 224
Sound Velocity Correction 225
View SVP positions 228
Navigation 229
Swath Cleaning 230
Swath Filtering 230
Apply Filters 233
Total Propagated Error 234
Compute TPE 235
TPE Filtering 236
Load Error Data 240
Update Backscatter 242
Single Beam Cleaning 243
Single Beam Filtering 243
Moving Average Filter 245
Depth 246
Beam-to-Beam Slopes 246

Apply Filters 246
Restart Cleaning 247
Delta Draft 249
GPS Tide 250
Merge 253
Quality Control 255
QC Report - Step 1 255
QC Report - Step 2 256
QC Report - Step 3 257
Quality Control Statistics 258
Side Scan Processes 259
Line Calibration 260

9 **Navigation Editor 261**
Navigation Editor Interface 262
Line Interpolation 265
Examine Navigation Data 267
Reject, Accept and Query Data 269
Reject Data 269
Accept Data 269
Query Data 270

10 **Attitude Editor 271**
Attitude Editor Interface 272
Display Options 274
Time 275
Scale Data 276
Vertical view 276
Other Display Options 276
Colour Display 277
Smooth Data 278
Cleaning Data 281
Reject data 281
Accept Data 282
Query data 283
Query line 283
Filter Sensor Data 284

11 **Sound Velocity Editor 287**
Overview 288
Open an Existing File 289
Depth Scale 289
Create a New SVP File 290

Options 291
Show/hide toolbar. 292
Show/hide status bar 292
Edit SVP Data 293
Create New Profile 295

12 **Swath Editor 297**
Overview 298
Swath Editor Interface. 299
Plan View 301
3-D View. 301
Side View 302
Rear View. 302
Profile View 302
Amplitude View. 302
Data Display in Views. 303
Swath Editor Display Options 304
Vertical Exaggeration 304
Specified Min-Max 305
Soundings 306
Colour By Depth 306
Examine Swath Data 307
Swath Coverage Outline 310
Changing Track Lines 311
Examine 3-D Data 312
Swath Cleaning. 317
Reject Data. 317
Reject Swaths 318
Accept Data 318
Designated Soundings 318
Query Data. 320
Auto Cursor Mode 321
Middle-Mouse Button Selection 321
The Refraction Editor 322

13 **Single Beam Editor 325**
Introduction 326
Display Options 327
Primary and Secondary Data 329
Change Sounding Depths. 330
Change Depths 330
Add Depths. 331
Single Beam Cleaning 332
Reject Data. 332
Accept Data 333

Query Data 333
Auto-Cursor Mode 334
Barcheck Function 335

14 **Tide Editor 337**
Introduction 338
Editing Tide Data 340
Create a New Tide File 342
Options 343
Exit the Tide Editor 345

15 **Field Sheets 347**
Overview 348
Create a Field Sheet 349
Open and Close Field Sheets 352
Set Field Sheet Properties 353
CARIS Field Sheet Editor 354
Tiling 355
Open and Close Tiles 356
Set Tile Properties 357
Tile Display Query 359
Query Tiles 361
Profiles 362
Create Profile 362
Display Properties 363
Export Profile 368
Display Profile 370
Selected Soundings 371
Sounding Colour Options 371

16 **Subset Editor 373**
Overview 374
Workflow 375
Subset Tiles 376
Subset Tile Properties 377
Subset Tile Tracking 379
Subset Editor Interface 380
Open Subset Editor 380
Subset Editor Tab 381
Toolbar 384
View Subset Information 384
Measure Distance 385
Colour Options 386

Size and Resize Subset 387
The 3-D View 389
The 2-D View 393
Close Subset Editor 396
Subsets and BASE Surfaces 397
Subset Cleaning 398
Reject 398
Accept 399
Designated Soundings 399
Outstanding 400
Examined 400
Query 400

17 **BASE Surfaces 401**
Overview 402
Range Weight 404
Uncertainty Weight 405
Swath Angle Weight 406
BASE Surfaces and Weighted Grids 407
Create BASE Surface 408
BASE Surface (Step 1) 408
BASE Surface (Step 2) 409
BASE Surface (Step 3) 410
View BASE Surfaces 412
Properties 413
View Surface Properties 413
Set Attribute Properties 414
Colour Map Editor 417
Colour Range Editor 419
Update BASE Surfaces 421
BASE Surface Commands 422
Open 422
Close 422
Recompute 423
Add to BASE Surface 423
Remove From BASE Surface 424
Interpolate BASE Surface 425
Delete 427
Finalize BASE Surface 427
Combine 429
New Layer 431
QC Report 432
Surface Filtering 436

18 CUBE Processing 437
Introduction 438
Generating CUBE Surfaces 441
CUBE Surface (Step 1) 441
CUBE Surface (Step 2) 442
CUBE Surface (Step 3) 443
CUBE Surface (Step 4) 444
Advanced Options 446
Hypothesis Editing 449
Nominate Alternative Hypotheses 452
Hypotheses Cleaning 454
Update CUBE Surface 455

19 Statistical Surface Cleaning 457
Overview 458
Tiling 458
Surface Cleaning 459
Creating Tiles and Cleaning Data 461
Query Options 464
Query Tile Surface 464
Query Tile Soundings 465
Tile Histogram 466

20 Lidar Data 469
Overview 470
Sounding Quality and Status Flags 470
Lidar Data in HIPS 472
Toolbar 473
Waveforms View 473
Selecting Data 475
Selecting Detected and Alternative Data 477
Cleaning Sounding Data 479
Reject Data 479
Accept data 479

21 Calibration 481
Overview 482
Workflow 483
Create Subsets in HDCS 484
Perform Calibration 486
Sample Calibration Procedure 488

22 **Side Scan Data 491**
Overview 492
Workflow 492
Side Scan Editor 494
Side Scan Editor Interface 495
Side Scan Editor window 496
Side Scan Editor controls 497
Toolbar 500
Navigate in Side Scan Editor 501
Zoom 503
Selection 504
Colour Mapping 507
Measure Distance 509
Measure Shadow 510
Edit Side Scan Data 510
Side Scan Editor and Swath Editor 513
Signal Display Window 515
Processing 518
Recompute Towfish Navigation 519
Digitizing Sonar Altitude 521
Automatic digitizing 522
Manual digitizing 524
Interpolate Selection 525
Slant Range Correction 525
Apply Slant Range Correction 526
Revert/Reapply 528
Beam Pattern Correction 529
Create Beam Pattern Correction File 529
Apply Beam Pattern Correction 533
Angle-Varying Gain 534
Time-Varying Gain 535
Despeckle 538
Contacts 539
Create Contacts 539
Editing Contacts 542
Recompute Contact Position 544
Mosaics 545
Create Mosaics 546
Create Mosaic - Step 1 546
Create Mosaic - Step 2 547
Create Mosaic - Step 3 549
Create Mosaic - Step 4 550
Create Mosaic - Step 5 551
Mosaic Attributes 552
Intensity Properties 553
Add To Mosaic 554
Remove From Mosaic 554
Recompute 555
Image Classification 555
SIPS Template Wizard 557

SIPS Template Wizard (Step 1) 557
SIPS Template Wizard (Step 2) 559
SIPS Template Wizard (Step 3) 560
SIPS Template Wizard (Step 4) 561
SIPS Template Wizard (Step 5) 562
SIPS Template Wizard (Step 6) 563

23 **Batch Processing 565**
Overview 566
Open Batch Processor 566
Running the Batch Processor 568
Open HBP File 568
New HBP File 568
Save/Save As 568
Select Process 569
Process Data 570
Batch Processor Tasks 571

24 **Create Publications 573**
Overview 574
Publication utilities 574
Open a Template 576
Apply Image Settings 577
Apply Scale Bar settings 578
Enter text values 579
Publication Viewer 580
Publication Viewer commands 580
Template Editor 583

25 **Export Data 585**
Introduction 586
Select Format (Step 1) 588
Select Files (Step 2) 589
BASE Surface To ASCII 591
BASE Surface to ASCII (Step 3) 591
BASE Surface To ASCII (Step 4) 592
BASE Surface to BAG 593
BASE Surface to BAG (Step 3) 593
BASE Surface to BAG (Step 4) 594
BASE Surface To Image 595
BASE Surface to Image (Step 3) 595
Contacts 597
Contacts (Step 3) 597

Contacts (Step 4) 599
GSF to CARIS Map/HOB 600
GSF to CARIS/HOB (Step 3) 600
GSF to CARIS (Step 4) 602
GSF to HOB (Step 4) 603
HIPS Tide to ASCII 604
HIPS Tide to ASCII (Step 3) 604
HIPS To ASCII 605
HIPS to ASCII (Step 3) 605
HIPS to ASCII (Step 4) 606
HIPS To ASCII (Step 5) 607
HIPS To CARIS Map 608
HIPS to CARIS (Step 3) 608
HIPS to CARIS (Step 4) 610
HIPS to CARIS (Step 5) 611
HIPS To GSF 612
HIPS to GSF (Step 3) 612
HIPS To HOB 613
HIPS to HOB (Step 3) 613
HIPS to HOB (Step 4) 614
HIPS To HTF 615
HIPS to HTF 616
HIPS to HTF 617
SIPS To UNISIPS 618
SIPS TO UNISIPS (Step 3) 618
Mosaic To Image 619
Mosaic to Image (Step 3) 619
Projection 620
Export 621

A **HIPS Status Flags 623**
Status Flags and Values 624

B **Generic Data Parser 627**
Introduction 628
Data Parsing Components 629
Configuration File 631
Open a Configuration File 631
New Configuration Files 632
Save a Configuration File 632
Raw Data Files 633
Data Types 635
File Header 636
File Date 636
Time Stamp 637

Sound Velocity 637
Depth 638
Navigation 638
Attitude/Navigation/Tide 639
Event 640
Generic Data Parser Wizard 641

C **Tide File Formats 647**
Overview 648
Tide files (*.tid) 649
COWLIS 650
NOS Preliminary Tide Data 651
NOS Final Tide 652
Tide Zone Definition Files 653

D **Contact File Formats 657**
Introduction 658
ContactLine 659
ContactSingle 660
ContactMulti 662
ContactMultiPoints 664

1

1. Introduction

Purpose of HIPS and SIPS

HIPS and SIPS are designed to receive bathymetric and side scan sonar data from a variety of industry standard data-logging formats. HIPS and SIPS contain tools for interactively and automatically identifying bathymetric outliers. Also, there is a wide range of proven and powerful algorithms for applying value-added processes to the data. The result of all these operations is a clean and final dataset that is ready for the creation of wide range of data products available within the HIPS and SIPS software and other integrated CARIS products.

- The HIPS and SIPS products share a common Windows interface containing editors for vessel, attitude, navigation, single beam bathymetry, swath bathymetry, side scan sonar, and other data sensors. The program also contains a georeferenced project view window for displaying the spatial location of these observed data with other georeferenced data. Examples include orthophotos, raster charts, CARIS vector map files, and S-57 Edition 3.1 data.
- The products also share a common application program called the CARIS Field Sheet Editor (FSE). This module performs the tasks necessary to generate finished field sheets that can be used for presentation. FSE can generate contours and make shoal soundings selections. It can also add cartographic data to the field sheet and define a customized plot layout.

Development Partners

The development of a modern and powerful bathymetric and side scan sonar processing system was made possible through the cooperation of several organizations:

- The Canadian Hydrographic Service (CHS)
- The Ocean Mapping Group of the University of New Brunswick
- The National Oceanic and Atmospheric Administration (NOAA)
- University of New Hampshire

Design Principles

HIPS and SIPS are the result of an effort to develop software tools that efficiently process large quantities of data being produced by modern bathymetric and side scan sonar systems. Underlying the design are a number of principles:

- Retain all collected information without thinning measured data.
- Ensure cleaned data is quantitatively correct.
- Measured quantities should not be unreasonably altered (e.g., depths should not be smoothed) during cleaning.
- Offer fast, automatic algorithms. This “power tools” approach can reliably clean large quantities of measurements in a short time where the nature of errors is well understood and where it can be automatically determined what measurements are in error.
- Allow the hydrographer to check or override the automatic tools using interactive methods. This can be useful when confronted with data whose errors are not easily modelled. As part of this approach, powerful visualization techniques let the hydrographer quickly spot potential problems.
- Supply statistical and reporting functions to allow the hydrographer to control the quality of cleaning.
- Provide a clean, intuitive Graphical User Interface (GUI) to make the cleaning procedure efficient, interesting and stress free.
- Develop a generic data structure for measurements, which can accommodate all known measurement types and which is efficient for the cleaning procedure.
- Efficiently transfer cleaned data to a hydrographic database, retaining all required information about the survey and its soundings.
- Provide a wide range of output products.

Licensing

HIPS and SIPS are divided into three distinct products, each with its own license. You can purchase the product with the functionality best suited to your situation. Below is a table that outlines each licensing level and the options that are included with each level.

Product	Supported Formats	Data Cleaning	Data Processing
HIPS Singlebeam	• Atlas • GSF • Hypack • Winfrog • generic ASCII data	• Interactive single beam depth cleaning • Automatic single beam spike filters • Subset area cleaning • Automatic TPE filtering	• tide zoning, • sound velocity corrections • batch processing
HIPS Multibeam Professional	HIPS Singlebeam plus • Elac • Furuno • LADS • Seabeam • Seafalcon • Simrad • UNBReson • XTF	HIPS Singlebeam plus • interactive swath cleaning • automatic swath filters • refraction repair, • integrated side scan display • 3-D subset area cleaning • statistical surface cleaning	HIPS Singlebeam plus • tide zoning, • sound velocity corrections • BASE surfaces
SIPS Professional	• Atlas • CMAX • Coda • EdgeTech • MarineSonics • Qmips • Seabeam • Seafalcon • SEGY • Simrad • XTF	• side scan viewing and cleaning • digitize towfish altitude	• Recompute towfish navigation • slant-range correction • mosaics • generate side scan contacts

All HIPS and SIPS products include the following features:

- vessel configuration
- tide/sound velocity preparation
- attitude/navigation cleaning
- GPS RTK tide
- background displays
- variable depth tiling
- contouring
- plotting

HIPS I/O Library

HIPS and SIPS processes hydrographic data from raw data to finished products by using a number of built-in applications. These applications meet the requirements for most hydrographic agencies. Some users, however, might wish to process the data using external or customized applications, including external data viewers and statistical algorithms. To satisfy these needs, a separate software development package called the HIPS and SIPS I/O Library can be purchased. This library is a collection of basic functions that let you extract data from the HIPS and SIPS format or create new datasets. You can also update existing files.

2

HIPS and SIPS Interface

Opening HIPS and SIPS

1. To open HIPS from the Windows interface, select **Start** > **Programs** > **CARIS** > **HIPS and SIPS 6.1** > **CARIS HIPS and SIPS**

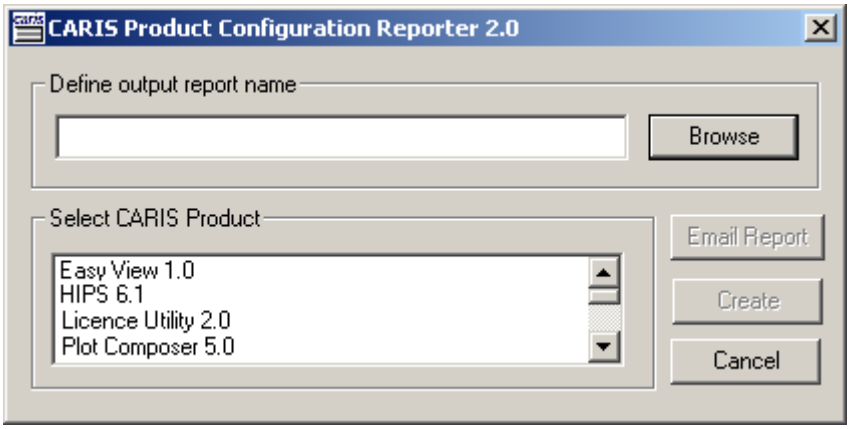
During installation you were prompted to place a HIPS icon on the desktop. If you selected this option you can then open HIPS and SIPS by double-clicking the icon.

Configuration Report

The configuration report is a text file containing a list of program, environment and system resources used by HIPS and SIPS.

1. From the Windows main menu, select **Start** > **Caris** > **Utilities** > **Configuration Reporter**.

The Configuration Reporter is displayed.



The CARIS Product Configuration Reporter contains a list of CARIS products already installed on your computer. This list is inactive until you select a path and name for the configuration report

2. Type a file path and name for the configuration report file, or click **Browse** and select a file path and type a name.

The list of CARIS products is now active.

3. Select HIPS and SIPS (version number) from the list.
4. Click **Create** to create a report in the selected folder, or click **Email Report** to create the report and send it as an email attachment to CARIS support.

The Configuration Report contains the following information:

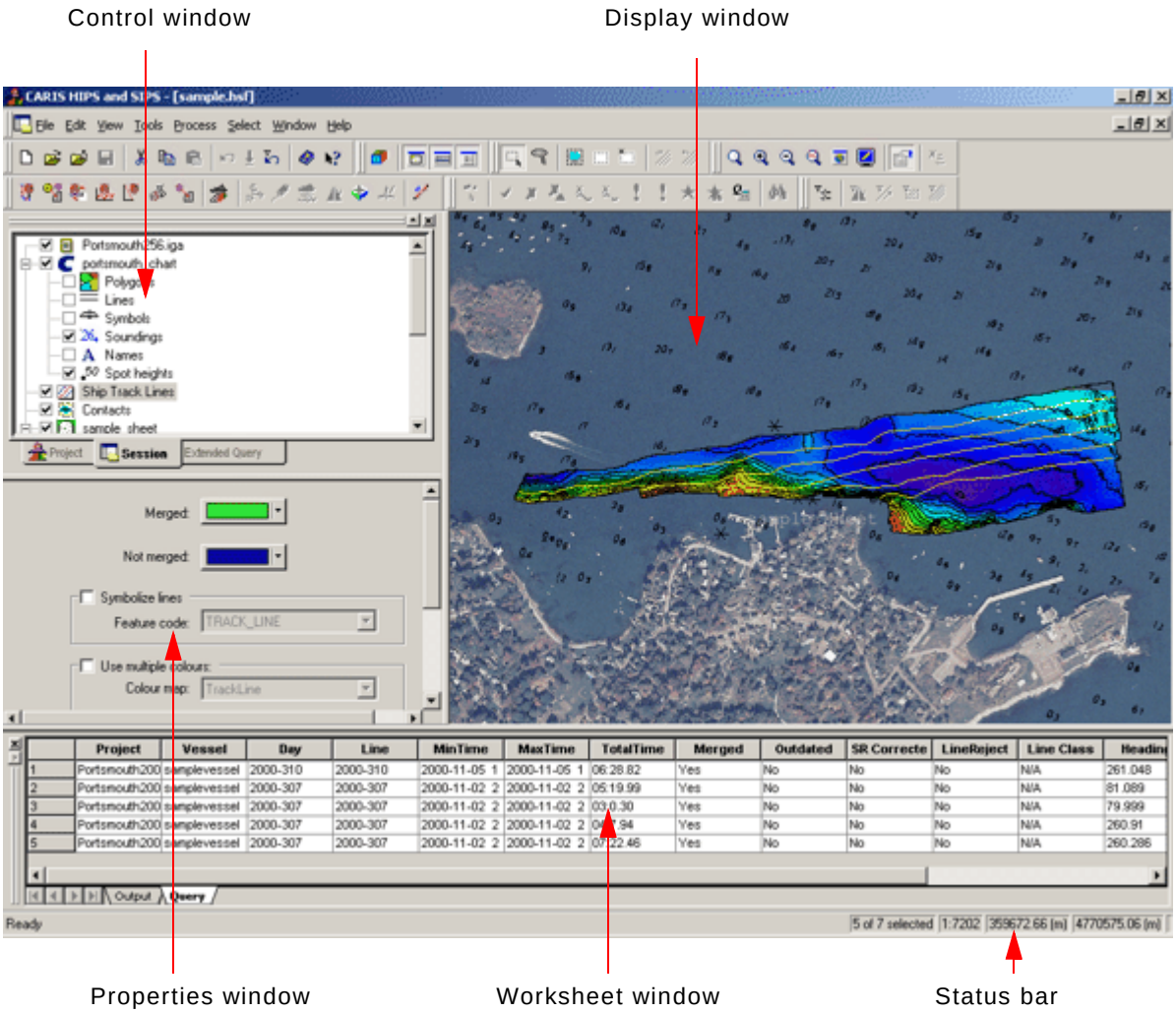
- version program files
- HIPS and SIPS environment settings
- Field Sheet Editor environment settings
- CARIS kernel registry
- system information
- system and user's environment

CARIS licensing report

Main Display

The interface has four windows:

- Display window
- Worksheet window
- Properties window
- Control window



The windows can be displayed, hidden, or resized and moved to another location (see "REPOSITION AND RESIZE WINDOWS" ON PAGE 22).

The interface includes toolbars (see "TOOLBARS" ON PAGE 24) and a status bar (see "STATUS BAR" ON PAGE 23).

Display Window

Menu	Window > Display
Tools	

The Display window shows a graphical representation of the survey area.

1. Select the Display window command.

You can perform the following functions in this window:

- select track lines (see "Select Survey Lines" on page 219)
- zoom and pan data (see "Zoom" on page 37 and "Pan" on page 39)
- open graphic files (see "Open Background Data" on page 31) and S-57 data (see "Display S-57 Data" on page 34)
- close track lines (see "Close a Project" on page 72)
- edit navigation data (see "Examine Navigation Data" on page 267).
- define a field sheet area (see "Create a Field Sheet" on page 349).
- save screen captures of the display in the window (see "Screen Captures" on page 47)

For options for the display and colour features for the ship and towfish track lines see "Display window" on page 78.

Control Window

The Control window is divided into three tabs: the Project tab, the Session tab, and the Extended Query tab.

1. Select the Control window command.

Menu	Window > Control
Tools	

The **Project tab** displays the currently open project files. Files are organized in a Project-Vessel-Day-Line hierarchy.

Expand a file tree by clicking the plus sign (**+**) beside a folder.

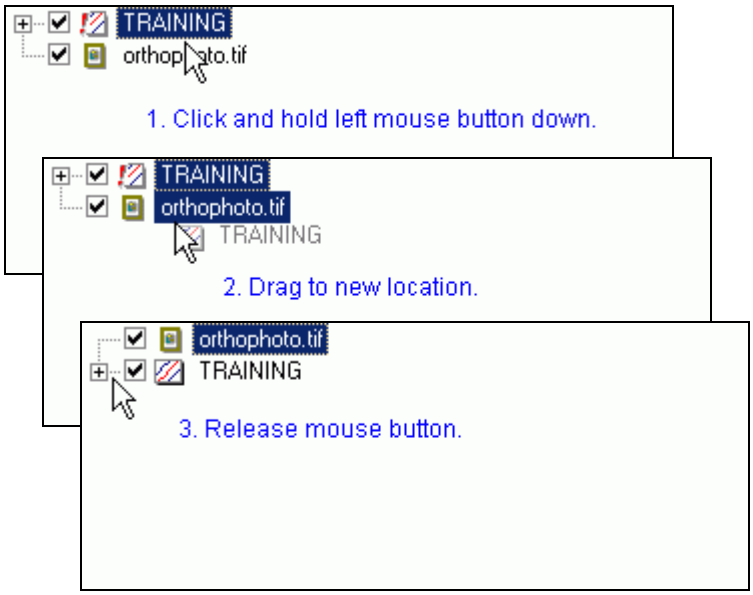
You can perform the following operations in this tab:

- open and close lines (see "CLOSE A PROJECT" ON PAGE 72)
- select track lines (see "SELECT SURVEY LINES" ON PAGE 219)

The **Session tab** lists layers that are open in the Session. By selecting or clearing layer check boxes, you can control what is viewed in the Display window.

Examples of layers include field sheets, BASE Surfaces, background data (CARIS maps and images), and field sheet products such as contours, selected soundings, tiles, etc.

The Session tab lists the order that objects are drawn to the Display window. Objects are drawn in order, from top to bottom, as they appear in the Session tab. You can change the draw order by dragging an object to another position on the list.



The **Extended Query** tab displays totals for the selected data. For example, the total rejected soundings for a line.

Editor tab

The Control window is also used in conjunction with HIPS and SIPS editors

You can perform the following functions in this window (depending on the editor that is open at the time):

- modify Swath Editor windows (see "Swath Editor" on page 297)
- repair refraction problems (see "The Refraction Editor" on page 322)
- display field sheet information (see "Open and Close Field Sheets" on page 352)
- display BASE surface information (see "Open" on page 422)
- filter attitude data (see "Filter Sensor Data" on page 284)
- examine navigation data (see "Examine Navigation Data" on page 267).
- control the display of subset data (see "Subset Editor Interface" on page 380)
- control the Side Scan Editor display (see "Side Scan Editor" on page 494)
- control contact information (see "Contacts" on page 539).

Properties Window

The Properties window controls the display properties of layers such as colour display, filtering options, etc. The Properties window can undocked and moved around the interface.

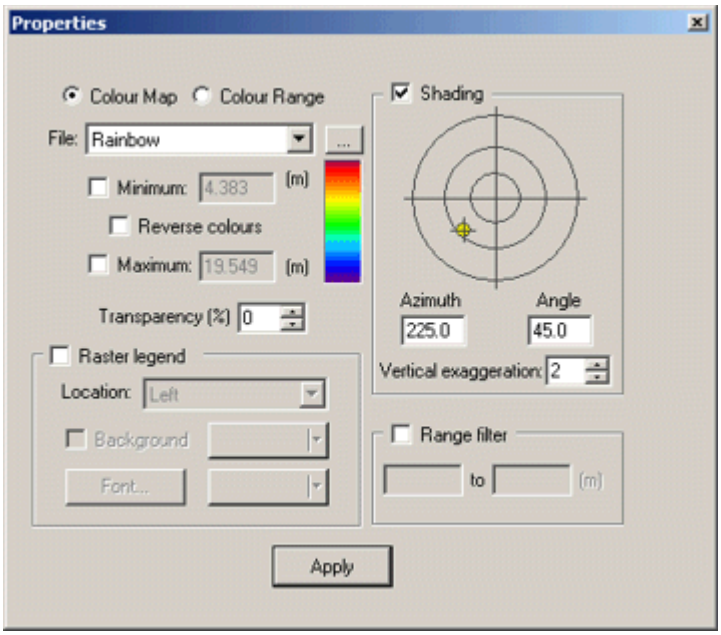
The display options shown in the window are determined by the selected layer type (not all layers contain display properties).

Properties window

Menu	Window > Properties
Tools	
Pop-up	Properties

1. Select the Properties window command.
2. Select a layer in the Session tab of the Control window.

The display properties for that layer are shown in the window.



Example of an undocked Properties window for a BASE Surface attribute layer.

3. Click **Apply** to implement the property changes.

See the following topics for more information on using the Properties window with specific layers.

- "Set Track Line Properties" on page 30
- "Set Image Properties" on page 32
- "Set Field Sheet Properties" on page 353
- "Set Tile Properties" on page 357
- "View Surface Properties" on page 413
- "Set Attribute Properties" on page 414

Worksheet Window

Menu	Window > Worksheet
Tools	

1. Select the Worksheet window command.

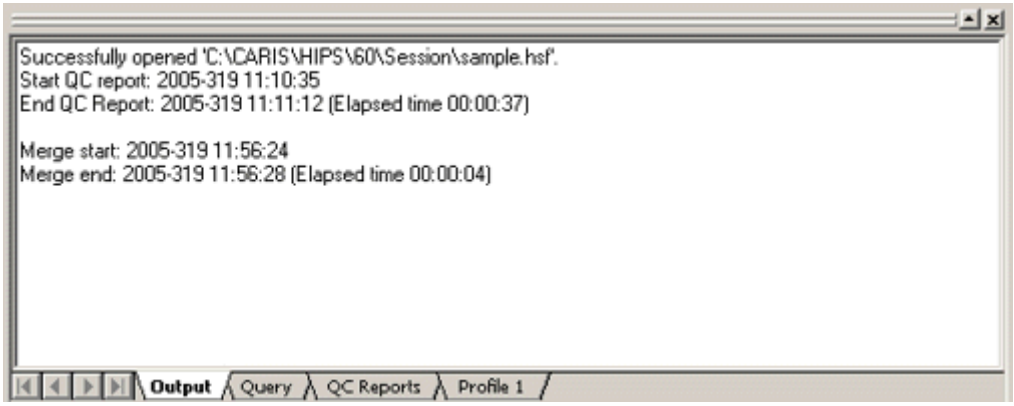
The Worksheet window is divided into five tabs:

- Query (for viewing information on selected data)
- Output View (for viewing the result of a command or process).
- Quality Control (for displaying quality control reports)
- Graphs (for displaying graph information)
- Profile (for displaying cross-sectional views of the BASE Surface or Tile area).

The Query and Output are visible all the time while HIPS and SIPS is open. The Quality Control, Graph, Profile tabs are only displayed as needed.

Output Tab

In this tab you can view the results of a command or process.

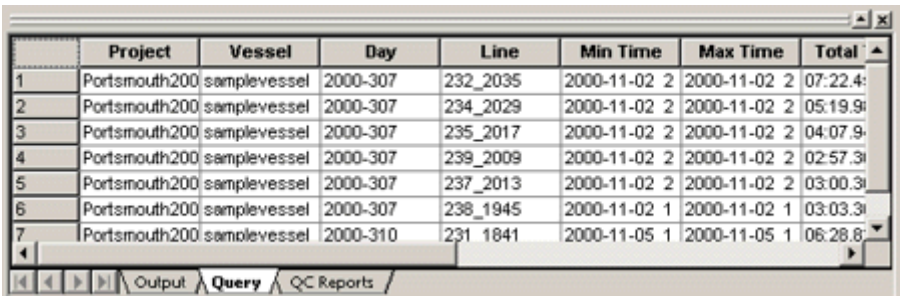


Information in this tab is read-only and cannot be modified.

If you want to remove the information in this tab, right-click the mouse and select Clear from the pop-up menu.

Query Tab

The Query tab is used for viewing selected data in the Display window.

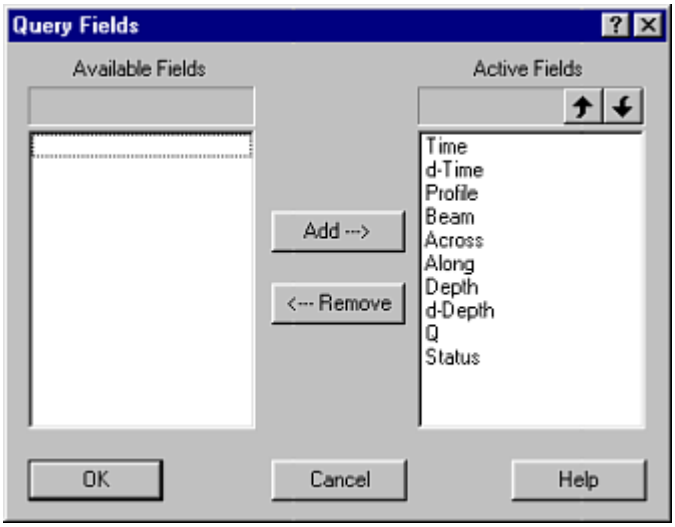


The data is shown in tabular format. You can invert the order of data by double-clicking a column header.

The type of data fields displayed in the Worksheet window and the order in which these fields are displayed is controlled by the Query Fields dialog box.

1. Right-click the *Query* tab in Worksheet window and select the Query Fields command from the pop-up menu.

The Query Fields dialog box is open.



The data types listed in the dialog box are determined by the editor currently open in the application.

2. To add data fields to the Worksheet window, select a data type in the *Available Fields* list and click the **Add** button.

The data field is now listed in the *Active Fields* list.

3. To remove a data field from the Worksheet window, select a data field in the *Active Fields* list and click the **Remove** button.

The data field is now in the *Available Fields* list.

4. To change the display order of data, select a data field in the *Active Fields* list and click the up or down arrow buttons.

The data field is now in a different position on the list.

5. Click **OK**.

The fields in the *Active Fields* list are visible in the Worksheet window. The order in which the fields are displayed is determined by the order in the *Active Fields* list.

Graph data

Data shown in the Query and QC Report tabs can also displayed in graphical format. See "GRAPH TAB" ON PAGE 18

Query options

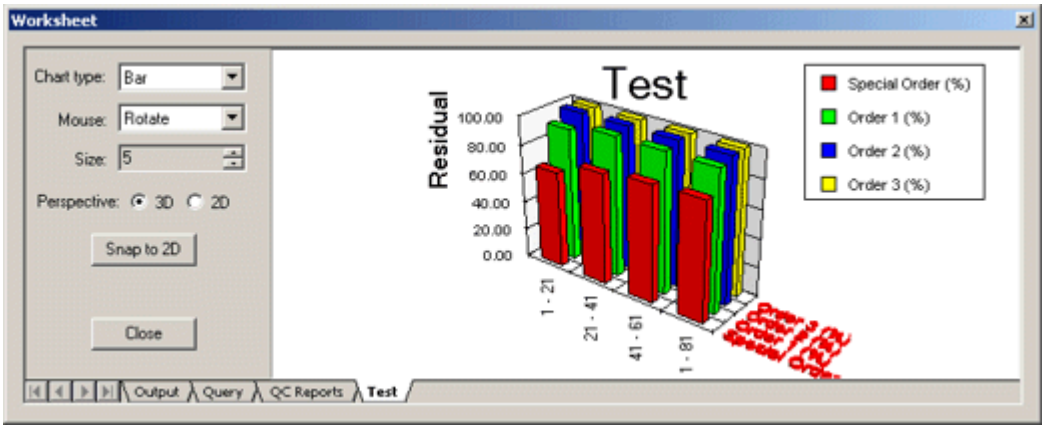
The following options are also available in the Query tab.

1. To save the data as a text file, position the cursor in the Query tab, right-click the mouse and select Save As from the menu.
2. To print the data, position the cursor in the Query tab, right-click the mouse and select Print from the menu.
3. To copy data, position the cursor over the field you want to copy (or right-click the mouse and use the Select All command to highlight all data) and use the Copy command.
4. To remove row(s) from the table, select the rows so they are highlighted, right-click the mouse in tab and select Remove Selected from the pop-up menu.

Menu	Edit > Copy
Tool	
Pop-up	Copy
Key	<Ctrl+C>

Graph Tab

The Graph tab displays tables in the Query and QC Reports tabs in graphical format.

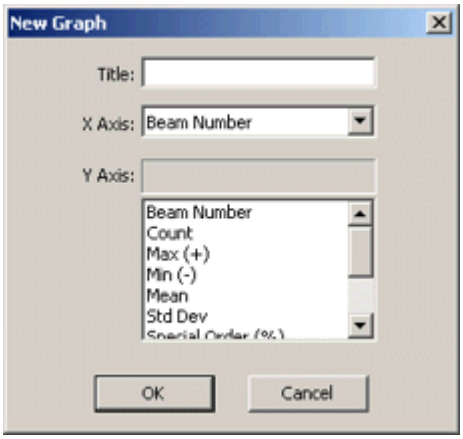


The graph can be rotated. Zoom and pan operations can also be applied. A legend is automatically generated when more than one data type is displayed.

Create Graph

1. While the cursor is in the Query and QC Report tabs of the Worksheet window, right-click the mouse and select New Graph from the pop-up menu.

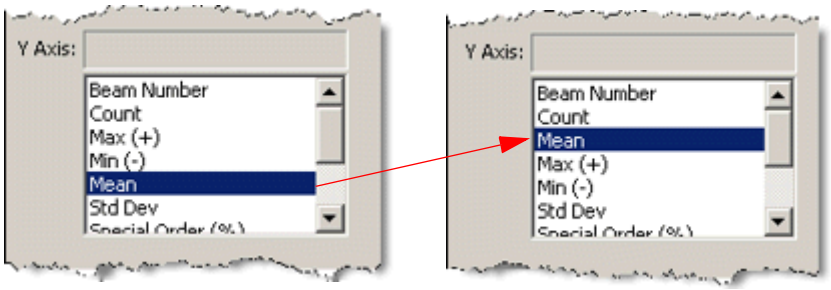
The New Graph dialog box is displayed.



The fields in the tab of the Worksheet window determine the available values for creating the x- and y-axes of the graph.

2. Type a name for the graph in the *Title* field.
3. Select values for the x- and y-axes. You can select more than one value for the y-axis.

Data is displayed in the graph according to the list order. If you select more than one data type for the y-axis, the first data type may hide the other types in the graph. You can avoid this by using the mouse to move a data type up or down on the list.

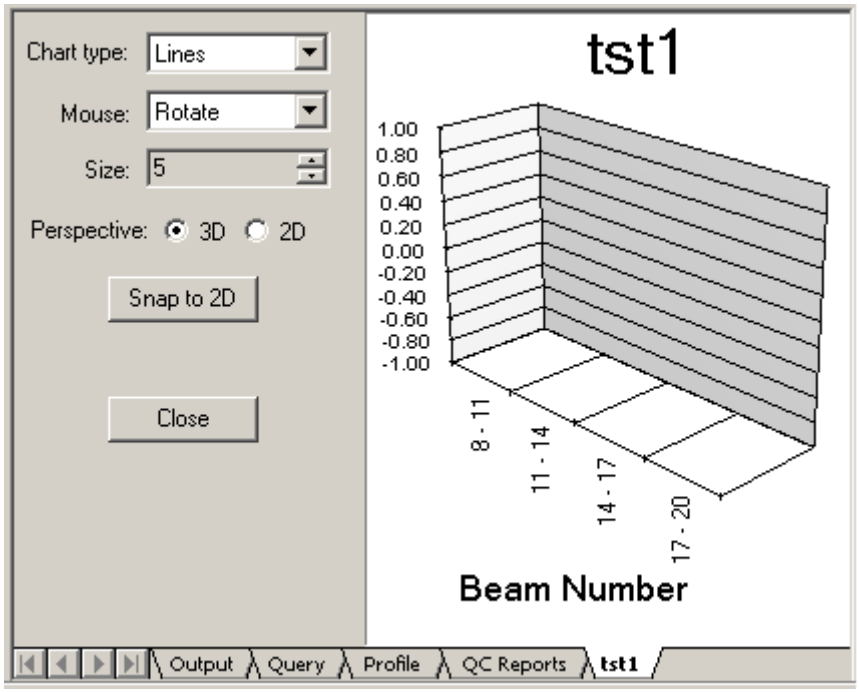


4. Select a data type, press and hold the mouse button while moving the data type to a new position on the list.
5. Click **OK**.

A graph is created in the tab.

Graph Controls

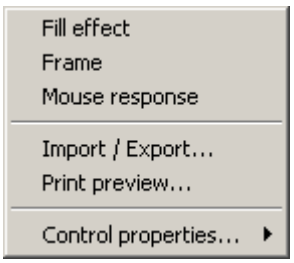
The display and control options are located on the left side of the tab.



1. Choose a display mode from the *Chart Type* drop-down list. Generally, you can select from points, lines or bars.
2. Graph positioning and other movement is controlled using the left mouse button. Select a mouse operation (rotate, pan, or zoom) from the *Mouse* drop-down menu.

3. Change the size of points, stars, diamond, crosses, diagonal crosses, and tubes
4. Select *3D* to display the graph showing depth, or select *2D* to display the graph in an orthographic display without depth.
5. To flatten the chart, click the **Snap to 2D** button.

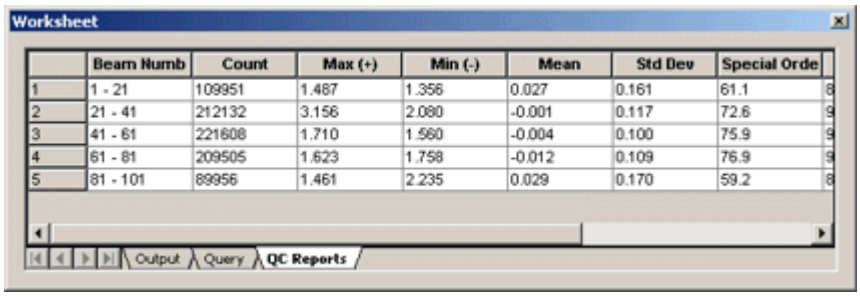
More detailed control graph control options are available from a pop-up menu in the graph area



A separate help file is available for these options.

QC Reports Tab

The QC Reports tab contains statistical data generated from the QC Reports tool. For more information on QC Reports, see "QUALITY CONTROL" ON PAGE 274.

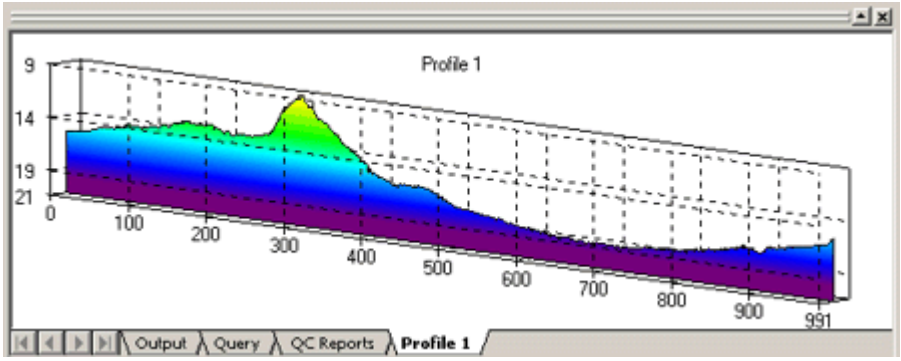


The same options are available from the QC Reports tab as in the Query tab, see "QUERY TAB" ON PAGE 16 for more information.

You can also create graphs using data from the table, see "GRAPH TAB" ON PAGE 18 for more information.

Profile Tab

View a cross-sectional diagram (profile) of a BASE Surface, weighted grid, or tile.



For more information on the options available in this tab, see "PROFILES" ON PAGE 362.

Reposition and Resize Windows

Repositioning

All windows can be undocked from their default positions and moved anywhere on the screen.

1. Position the cursor at the top of the window, or double-click the handle.
2. Press and hold the mouse button and drag the cursor.

As you drag the cursor, the window un-docks from its current location and moves with the cursor. To prevent a window from automatically docking, hold the <Ctrl> key while dragging the window.

3. Release the mouse button when you want to dock the window in its new location.

Resizing

Windows can also be sized to a new length and width.

1. Position the cursor on any edge of the window.

The cursor becomes a two-headed arrow.

2. Press and hold the mouse button and use the cursor to drag the edge of the window to a new position.

As you drag the cursor, the edge of the window expands or contracts, depending on whether you are pulling or pushing the window edge.

3. Release the mouse when you resized the window to the desired length and width.

Restore default windows layout

This command will close all open editors and reset the interface windows to the same appearance as when the application was first installed.

You will be prompted to save changed data as editors are closed.

1. Select the Restore Default Window Layout command.

Menu	Window > Restore Default Window Layout

Status Bar

The status bar at the bottom of the interface displays the following information:

- message about the operation currently taking place, or the information in the tool tip for the command button or menu item your cursor is on
- number of track lines selected out of the total lines open in the interface
- scale of the current display in the Display window
- coordinates of the current cursor position in the units chosen in Tools > Options
- disk usage for a selected data repository

Depending on which editor you have open the status bar will also display information such as the day the survey was conducted, profile and beam number, etc.

Status Bar options

You can hide the status bar:

1. Select the Status Bar command.

Menu	View > Status Bar

The check mark beside the **Status Bar** option is gone and the status bar is no longer visible in the desktop.

Cursor coordinates

The units for the coordinates of the cursor's current position are set in the Options dialog box of the Tools menu. You can use the Units option to set the type of unit e.g. Geographic or Ground.

The Position Precision option is used to set the number of places to the right of decimal point in the coordinate display.

For further information see "UNITS" ON PAGE 95.

Toolbars

Toolbars allow swift access to commonly used commands. Toolbars can undocked and moved about the interface by grabbing the handle on the left side of the toolbar with the cursor. When moving a toolbar, press and hold the <Ctrl> key to stop the toolbar from automatically docking.

The following is a list of toolbars.

- **Standard:** Access standard Windows commands such as opening and saving files.
- **Edit:** Access functions used for editing data.
- **Window:** Access commands for toggling windows to on or off.
- **View:** Access functions for controlling the view in the Display window.
- **Tools**: Access functions to open editors and functions used for processing data.
- **Process:** Access functions for executing processes such as creating BASE surfaces and merging.
- **Filter:** Access functions defining and executing automatic filters.
- **Select:** Access functions for selecting objects in the Display window.
- **Swath Editor:** Access functions for examining and cleaning pre-merged sounding data.
- **Single Beam Editor:** Access functions for examining and cleaning single-beam data.
- **Subset Editor:** Access functions for examining and cleaning sounding data in the subset mode.
- **Side Scan Editor:** Access functions for examining data, slant-range correcting raw side data, and creating contacts.
- **Lidar:** Access functions for selecting and examining soundings recorded by a lidar system.
- **Surface Cleaning:** Access functions for surface cleaning and tiling.

You can also create a custom toolbar (see "CREATE A CUSTOM TOOLBAR" ON PAGE 26).

Modify Existing Toolbars

Most toolbars are visible when the application is opened. You can remove toolbars from the desktop.

Menu	View > Toolbars

1. Select the Toolbars command.
2. Select the a toolbar name from the Toolbars sub-menu so it is no longer checked.

The toolbar is removed from the desktop.

To make a toolbar visible again, select the toolbar name from the sub-menu so it is checked.

Move toolbars

Toolbars can be moved to any location on the desktop.

1. Position the cursor over any area of the toolbar not covered by a button, or double-click the double bar grab handle on the left side of the toolbar.
2. Press and hold the mouse button while dragging the toolbar to a new location.

Modify appearance

You can alter the buttons' size and appearance.

1. Select the Customize command.

Menu	View > Toolbars > Customize

The Customize dialog box is displayed.

2. Select any of the following three options by checking a box.
   - *Tool Tips*: Display the name of a command button when the cursor is placed over it.
   - *Cool Look*: Remove the border outline from the buttons.
   - *Large Buttons*: Display buttons with 32 x 32 pixel icons instead of 16 x 16 pixel icons.
3. Click **OK**.

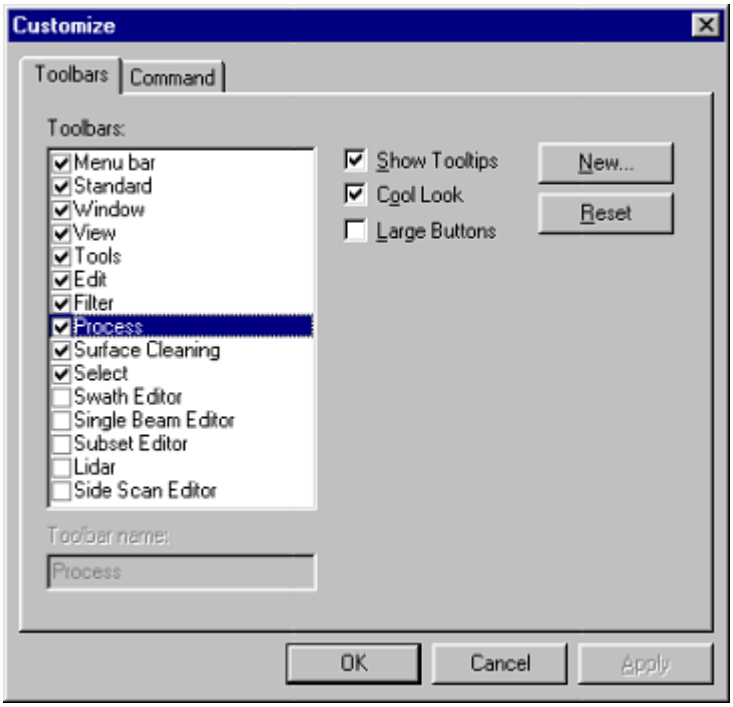
Create a Custom Toolbar

Create a custom toolbar by combining commonly used buttons from other toolbars.

1. Select the Customize command.

Menu	View > Toolbars > Customize

The Customize dialog box is displayed.



2. Click the Toolbars tab.
3. Click **New**.

The New Toolbar dialog box is displayed.

4. Type a name for the toolbar.
5. Click **OK**.

A new toolbar is displayed in HIPS/SIPS.

6. Click the *Commands* tab.
7. Select a toolbar name in the *Categories* list.

The buttons associated with that toolbar are displayed on the right side of the dialog box.

8. Select a button so it is highlighted.
9. Press the mouse button and drag the highlighted button to the new toolbar.
10. Repeat Steps 7 to 9 until you have all the required buttons.

Delete a toolbar

Remove a new toolbar.

> The Delete option can only be applied to new toolbars and cannot be applied to default toolbars.

Menu	View > Toolbars > Customize

1. Select the Customize command.

The Customize dialog box opens.

2. Select the *Toolbars* tab (if the tabbed window is not already visible).
3. Select the new toolbar's name in the *Toolbar* section, so the name is highlighted.

The **Reset** button becomes a **Delete** button.

4. Click the **Delete** button.

The toolbar is removed from the desktop and is no longer listed in the Customize dialog box.

Mouse Information

	Mouse button use varies according to the location of the cursor in the interface. Below is an overview of the most important functions.
Left button	The left mouse button is used for general manipulation of the interface, such as placing cursor in text fields, selecting objects in the Display window, selecting menu commands, and selecting data in the editor views.
Right button	The right mouse button displays context sensitive pop-up menus in any of the windows.
Middle button	The middle mouse button is used to pan the display using a press and drag action. Note that while panning the cursor changes to a four-pointed arrow.
Mouse wheel	The mouse wheel can be used to zoom in or out of Display window. It can also be used to move along a track line when an editor is active (Attitude Editor, Swath Editor, Side Scan Editor, etc.).

Open Projects and Background Data

Open a Project

Menu	File > Open Project
Tool	
Key	<Ctrl + O>

1. Select the Open Project command.

The Open Project dialog box is displayed.

2. Select a Project folder so it is highlighted.

If the project contains both bathymetric and side scan data—you can open either the vessel or towfish navigation, or open both. The setting is saved in the registry and the session file for future use. This option is inactive while a project is open.

3. Select either the *Side Scan Only*, *Ship Only* or *Both* options to open the appropriate track lines in the Display window.
4. Click **Open**.

The track lines associated with the project are opened in the Display window. The Project/Vessel/Day/Line file tree is displayed in the Control window.

Open individual track lines

Specific track lines can also be opened in Display window.

1. Choose the Open Project command.

The Open Project dialog box is displayed.

2. Select a Project folder so it is highlighted.
3. Expand the folder tree by clicking the + icon so that the Vessel and Day folders are visible.
4. Select a Day folder.

The Line folders are displayed in the Contents window of the dialog box.

5. Select a line folder so it is highlighted.
6. Select either the *Side Scan Only*, *Ship Only* or *Both* options to open the appropriate track lines in the Display window.
7. Click **Open**.

Combining project data using Windows shortcuts

You can use a Windows shortcut to link HDCS Project\Vessel\Day\Line data on different drives or network locations. The project, plus the shortcut to the other data, is displayed in the Open Project dialog box. The data folders must contain a Vessel Configuration File.

1. Create a Windows shortcut in any P\V\D\L directory.
2. Copy the shortcut to the project where you want to place the shortcut. Remember that the shortcut must be copied to the appropriate folder. A Project folder shortcut must be copied to a HDCS data folder, a Vessel folder shortcut must be copied to a Project folder, etc.
3. Open the data in using the Open Project command.

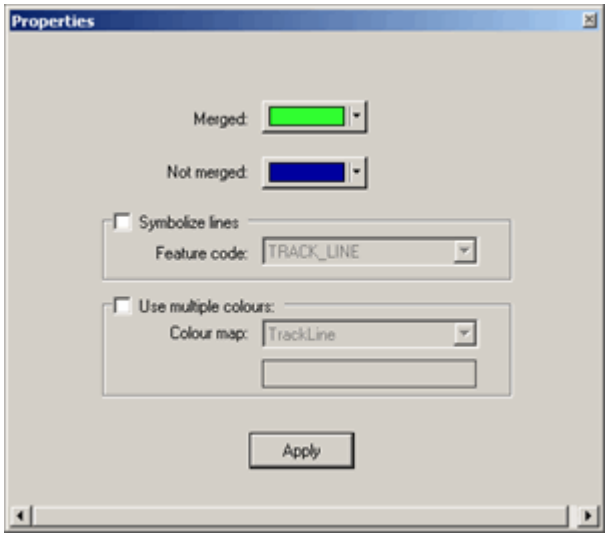
Set Track Line Properties

Change the colour and symbology of ship and towfish track lines.

1. Select the ship or towfish track lines layer in the Session tab of the Control window.
2. Select the Properties window command if the window is not open, or right-click the layer and select Properties from the pop-up menu.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window for ship or towfish track lines is displayed. The following window is for ship track lines, but the Properties window for towfish track lines is similar.



Ship track lines can be coloured by whether data has been *Merged* (combined into a position/depth file) or *Not Merged*.

Tow fish track lines can be coloured as *Slant-Range Corrected* or *Raw Side Scan*.

3. Select a colour from the colour palette, or create a custom colour from the standard Windows colour picker.

The following steps are applicable to both ship and towfish track lines.

4. Select the *Symbolize Lines* check box to add symbolization according to a selected feature code.
5. Select a *Feature Code* for line symbolization from the drop-down list.

The *Use Multiple Colours* option uses a set of 10 distinct colours to colour the track lines in the Display window. This is useful if the data was logged as multiple files along the same runline.

6. Select the *Use Multiple Colours* check box to implement this option and select a colour map from the drop-down list.
7. Click **Apply**.

Open Background Data

HIPS and SIPS can open any of the following data types as background in the Display window:

- AutoCAD files (DXF, DWG)
- geo-referenced TIFF (GeoTIFF, TIFF/TFW, TIFF/IGA)
- JPEG 2000 (JP2)
- Shapefiles (SHP)
- raster charts (BSB, HCRF)
- CARIS maps (DES)
- S-57 Ed 3.1[1] (000)
- Hydrographic Object Binary (HOB) files (including Edit and Marker layers created in CARIS Notebook).
- MrSid image files (SID)[2]
- Bathymetric Attributed Grid files (BAG)

1. Select the Open Background Data command.

Menu	File > Open Background Data

The Open dialog box is displayed. More than one data type can be selected while the dialog box is open.

1. Update files included with the .000 file are automatically opened with the original S-57 file
2. MrSID image files can only be opened if you have installed the CARIS plugin for MrSID images. This free plugin can be downloaded from http://www.caris.com/products/mrsid.cfm.

2. Select a file or group of files by one of three methods:
   - Select an individual file by highlighting it.
   - Select a group of files by pressing the Ctrl key and selecting each file.
   - Select a range of files by pressing the Shift key and selecting the first and last file in the range. All files between the first and last file are highlighted.
3. Click **Open**.

The files are opened in the Display window and are listed as layers in the Control window.

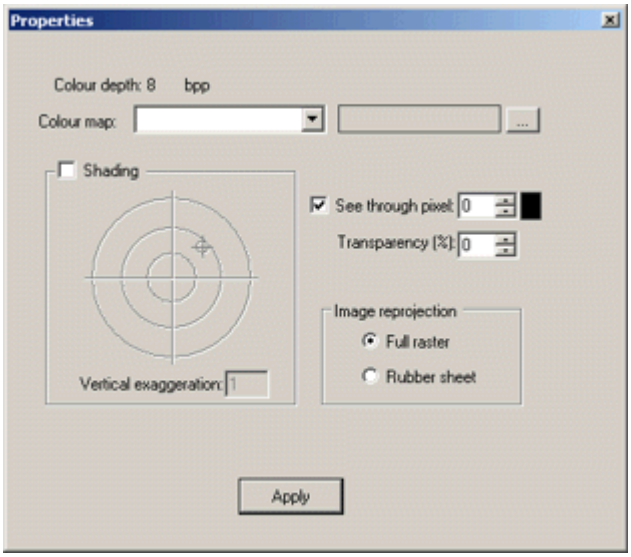
Set Image Properties

Set the display options for image files (TIFF, TWF, IGA, raster images etc.).

1. Select the Image layer in the Session tab of the Control window.
2. Select the Properties window command if the window is not open, or right-click the layer and select Properties from the pop-up menu.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window is displayed.



3. Select a colour map for a raster image, if applicable, from the drop-down list. A preview of the colour map is displayed to the right of the list.

The Shading and Vertical Exaggeration options are applied to GIS raster images where height values are available.

The *Shading* option controls the lighting angles of the raster image. Using this option may highlight fine scale features in the image that would not normally be seen.

The sundial control sets the shading effect in the image. The elevation varies from 90 to 0 degrees starting from the middle of sundial. The azimuth varies from 0 to 360 degrees in a clockwise direction.

4. Select the *Shading* check box to activate the sundial and *Vertical Exaggeration* options.
5. Position the cursor over the button in the sundial and press and hold the mouse button to drag the button to a new position.
6. Change the exaggeration of the illuminated sea floor by selecting a *Vertical Exaggeration* value. Increasing the exaggeration may improve the view of finer features in the image.
7. Select the *See Through Pixel* option to make a pixel value totally transparent. Type the value in the field.
8. To choose a transparency level for the entire image, click the up or down arrow buttons to select a *Transparency* percentage.

You can re-project a georeferenced raster image to a project projection using two methods:

- The *Full Raster* option is a pixel-by-pixel reprojection to the project coordinate system. This option can take a long time if the image file size is large.
- The *Rubber Sheet* option uses the centre coordinate in the Display window as a starting point for image reprojection. This is done by aligning the same coordinate in the georeferenced image to the centre coordinate in the window. The raster image is then reprojected out from the centre coordinate. This means that the image is reprojected every time you pan or zoom the Display window.

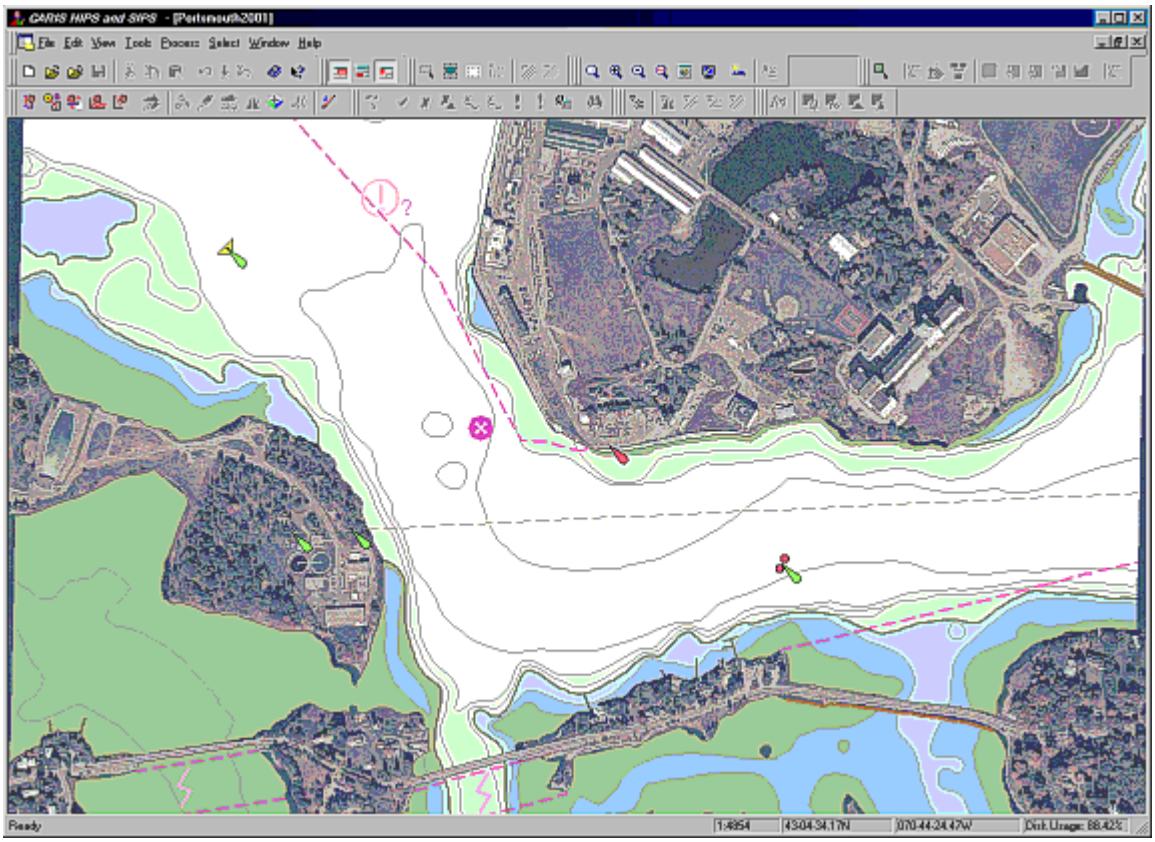
9. Select either the *Full Raster* or *Rubber Sheet* options.
10. Click **Apply** to implement the changes.

Display S-57 Data

HIPS and SIPS is capable of displaying S-57 Ed 3.1 and HOB (Hydrographic Object Binary) files. How the data is displayed is controlled by the settings in the Options dialog box (see "S-52 Display" on page 103) and Object Filter controls.

1. Select the Open Background Data command (see "Open Background Data" on page 31) and open an S-57 dataset.

The S-57 dataset is opened in the interface and the file is listed in the Session data tree in the Control window.

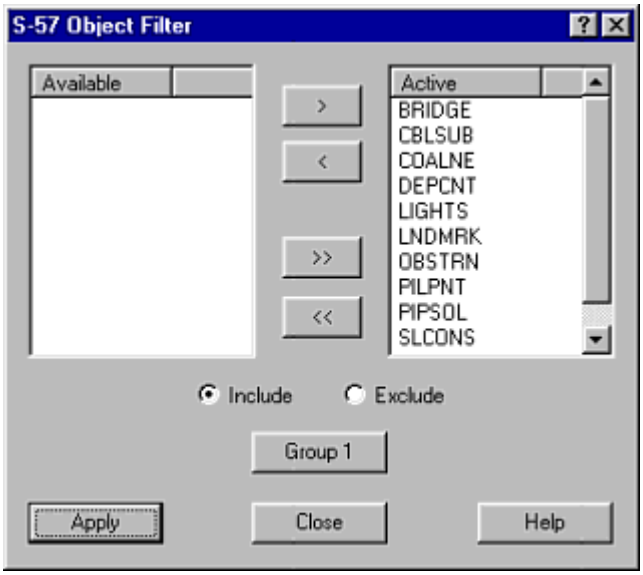


You can determine what objects are shown in the Display window.

1. Select the S-57 file in the *Session* tab of the Control window.
2. Right-click the mouse and select the Object Filter command.

The S-57 Object Filter dialog box is displayed.

Menu	View > Object Filter
Pop-up	Object Filter



The *Available* list contains all objects in the S-57 file. The *Active* list contains the S-57 objects that are currently shown in the Display window.

3. Move objects between the lists, as needed.
   - To move objects from the *Active* list to the *Available* list, select the object(s) and click the left-arrow button.
   - To move objects from the *Available* list to the *Active* list, select the objects and click the right-arrow button.
   - To move all objects from the *Active* list to the *Available* list, click the double left-arrow button.
   - To move all objects the from the *Available* list to the *Active* list, click the double right-arrow buttons.
   - To move all Skin of the Earth objects to the *Active* list, click **Group 1**.
4. Select the *Include* or *Exclude* options to show or hide objects in the Active list.
   - Select the *Include* option to show objects in the Active list.
   - Select the *Exclude* option to hide objects in the Active list.
5. Click **Apply** to save the settings.

Data about selected S-57 objects can be displayed in the Worksheet window using the Query command.

1. Select the S-57 file name in the Control window.
2. Select the S-57 objects in the Display window that you want to query.

Menu	Edit > Query
Tool	
Key	<Q>

3. Select the Query command.

Information about the selected S-57 objects is displayed in the Worksheet window.

Further information about an object (if available) can be accessed by selecting the *Extended Query* tab.

4. Select an object and select the *Extended Query* tab.

Display Tools

Zoom

The Zoom tool lets you enlarge or reduce the viewing area in the Display window.

Menu	View > Zoom
Tool	
Key	<F10>

1. Select the Zoom command.

The cursor changes to a magnifying glass icon.

2. Press and hold the mouse button and drag the cursor diagonally across the area of the Display window you want to enlarge.

The display is enlarged or reduced according to the zoom factor in the General Options dialog box (see "GENERAL" ON PAGE 75). The window is recentred on the location you selected.

Once the area has been zoomed, the Zoom option is no longer active. To keep the Zoom tool active, select the *Enable Constant Zoom* function in the Options dialog box (see "GENERAL" ON PAGE 75).

Zoom In

Enlarge the viewing area in the Display window.

Menu	View > Zoom In
Tool	
Key	<Ctrl+Down>

1. Select the Zoom In command

The display is automatically enlarged by the zoom factor set in the General tab of the Options dialog box (see "GENERAL" ON PAGE 75).

Zoom Out

Reduce the viewing area in the Display window.

1. Select the Zoom Out command.

Menu	View > Zoom Out
Tool	
Key	<Ctrl+Up>

The display is automatically reduced by the zoom factor set in the General tab of the Options dialog box (see "GENERAL" ON PAGE 75).

Zoom/Pan Previous

Go to the previous zoom/pan view.

1. Select the command.

Menu	View > Zoom/Pan Previous
Tool	

Zoom/Pan Forward

Go the zoom/pan view that was displayed prior to using the Reverse command.

1. Select the Zoom/Pan Forward command.

Menu	View > Zoom/Pan Forward
Tool	

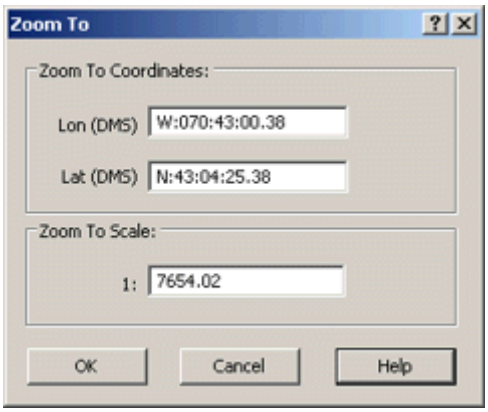
Zoom To

Recentre the view in the Display window and zoom into an area.

1. Select the Zoom To command.

Menu	View > Zoom To

The Zoom To dialog box is displayed.



2. Type the new coordinates in their respective fields to recentre on an area of the Display window.
3. Type a new scale to zoom into or out of the display.
4. Click **OK**.

Pan

The Pan tool allows you to move up, down or across the current display in the Display window.

Menu	View > Pan > Up/ Down/Left/Right
Tool	

1. Select a Pan command:
   - *Pan Up*: Recentre the display at a position that is above the current display area.
   - *Pan Down*: Recentre the display at a position that is below the current display area.
   - *Pan Left*: Recentre the display at a position that is left of the current display area.
   - *Pan Right*: Recentre the display at a position that is right of the current display area.

Refresh Display

Redraw the view in the Display window.

1. Select the Refresh command, or click the middle mouse button when the cursor is in the Display window.

The display is redrawn.

Menu	View > Refresh
Tool	
Key	<F5>

Overview Display

Redraw the Display window to the full extent of all the data that is currently open.

1. Select the Overview command.

Menu	View > Overview
Tool	
Key	<F9>

Show/Hide Layers

Layers are objects that can be viewed in the Display window. Examples of layers include CARIS maps, images, and field sheet products such as contours, selected soundings, tiles, etc.

You can show or hide objects in the Display window.

1. Make sure the Control window is open.
2. Select the *Session* tab if it is not already displayed.

The *Session* tab contains a list of objects available for display in the Display window.

3. If necessary, expand an object file tree by clicking the + icon beside it.
4. Show/hide objects by selecting or clearing the check box beside the object file name.
5. Select the Refresh command, or make sure the Automatic Refresh option in the General Options dialog box is selected (see "GENERAL" ON PAGE 75).

The object is displayed or hidden.

Menu	Window > Control
Tool	

Menu	View > Refresh
Tool	
Key	<F5>

Change draw order

Objects are drawn in order from top to bottom as they appear in the *Session* tab.

You can change the draw order by moving the object to another position on the list.

1. Select an object in the Control window so it is highlighted.
2. Press and hold the mouse button, and use the cursor to drag the object to another position in the list.
3. Release the mouse button when the object is moved.

The object is now shown in its new location in the data list.

Selection Tools

You can select an individual track line or object in the display by clicking on it with the mouse so it is highlighted.

While you can select a group of objects by holding down the <Ctrl> key while clicking on each object in turn, the following tools make selection of multiple objects easier.

Select by Range

Select one or more objects.

1. Select a layer in the Control window.
2. Choose the Select by Range command.
3. Press and hold the mouse button while dragging the cursor to define a rectangular area.

Menu	Select > By Range
Tool	

The selected objects from the layer are highlighted.

Select All in Display

Select objects that are totally or partially in the current viewing extent of the Display window.

This command is useful for selecting a single object or group of objects in an enlarged section of the Display window.

1. Select a layer in the Control window that contains the objects you want to select.
2. Use the zoom tool to enlarge a section of the Display window (see "ZOOM" ON PAGE 37).
3. Choose a Select All in Display command.

Menu	Select > All in Display
Tool	

Objects that cross or are within the current view of the Display window are highlighted.

Select All

Select all objects.

Menu	Select > Select All
Tool	
Key	<Ctrl+A>

1. Select a layer in the Control window that contains the objects you want to select.
2. Choose the Select All command.

All objects are highlighted.

Select By Lasso

In the Swath Editor, Subset, Navigation and Attitude Editors, use the By Lasso tool to select objects by enclosing an area.

The area is defined by the mouse in one of two ways: either by dragging the mouse around the area to be selected or by inserting a series of points to create an outline. All objects inside the defined area are selected when the lasso is closed.

Use Lasso in HIPS and SIPS editors

Menu	Select > By Lasso
Tool	

1. Select the Lasso command.
2. Click and hold down the left mouse button, and drag the mouse in any of the editor windows. The lasso outline is updated as you move the mouse.

   OR
3. Repeatedly click the left mouse button in the editor window to define a point-to-point outline.

Note: These two nodes can be used together in the editor windows.

To complete the selection area, you must close the lasso outline. This can be done in one of three ways:

- release the left mouse button and right-click.
- select Close Lasso from the Select menu
- press the “C” key.

Menu	Select > Close Lasso
Key	<C>

The first and last points will be connected and the lasso shape will be closed. The objects within the enclosed area are selected.

Use Lasso in Display window

To use the lasso tool to select objects in the Display window

1. Select the layer in the Control window that contains the objects you want to select.
2. Select the Lasso command.
3. Use the mouse to insert a series of points around the area in the Display window that you want to select.

The points are connected by a line as you continue clicking. When you have outlined the area

4. Press “C” to close the lasso outline.

Select Next Line

Select track lines by ascending file order. This command is enabled only when the Attitude, Navigation, Swath and Single Beam Editors are open.

The file order can be sorted according to time or name by right-clicking in the Display window and selecting **Sort** > **By Name**/ **By Time** from the pop-up menu.

1. Select the Next Line command.

Menu	Select > Next Line
Tool	

The track line directly below the currently selected line in the project data tree is highlighted.

Select Previous Line

Select track lines by descending file order. This command is enabled only when the Attitude, Navigation, Swath and Single Beam Editors are open.

The file order can be sorted according to time or name by right-clicking in the Display window and selecting **Sort** > **By Name**/ **By Time** from the pop-up menu.

1. Select the Previous Line command.

Menu	
Tool	Select > Previous Line

When you select the command, the track line directly above the currently selected line in the project data tree is highlighted.

Select Lines From the Control Window

Menu	Window > Control
Tool	

1. Make sure the Control window is open.
2. Select the Project tab.
3. Click a track line file so it is highlighted.

The track line in the Display window is highlighted.

You can also select more than one file.

- To select a Project, Vessel or Day folder, click the folder. All files in that folder are highlighted.
- To select multiple files, press and hold the <Ctrl> key while clicking the files with the mouse.
- To select a range of files, click the first file in the range, press and hold the <Shift> key while clicking the last file in the set. This highlights all adjacent files between the first and last file.

Clear All

Deselect track lines or objects.

Menu	Select > Clear All
Tools	

1. Select the Clear All command, or click anywhere in the display away from the lines.

The previously selected track lines or objects are no longer highlighted.

Screen Captures

Menu	View > Save Image
Pop-up	Save Image

You can take a screen capture of the current display in the Display window or 3-D Display window and save it as an 8-bit or 24-bit GeoTiff image.

1. Right-click in the Display window or 3-D Display window and select Save Image command.

The Save As dialog box is displayed.

2. Select a directory for saving the image.
3. Type a name for the image in the *File Name* field.
4. Select to either save the image as an 8-bit or 24-bit image by selecting the appropriate option.
5. Select a *Dots Per Inch* (DPI) setting from the drop-down list.

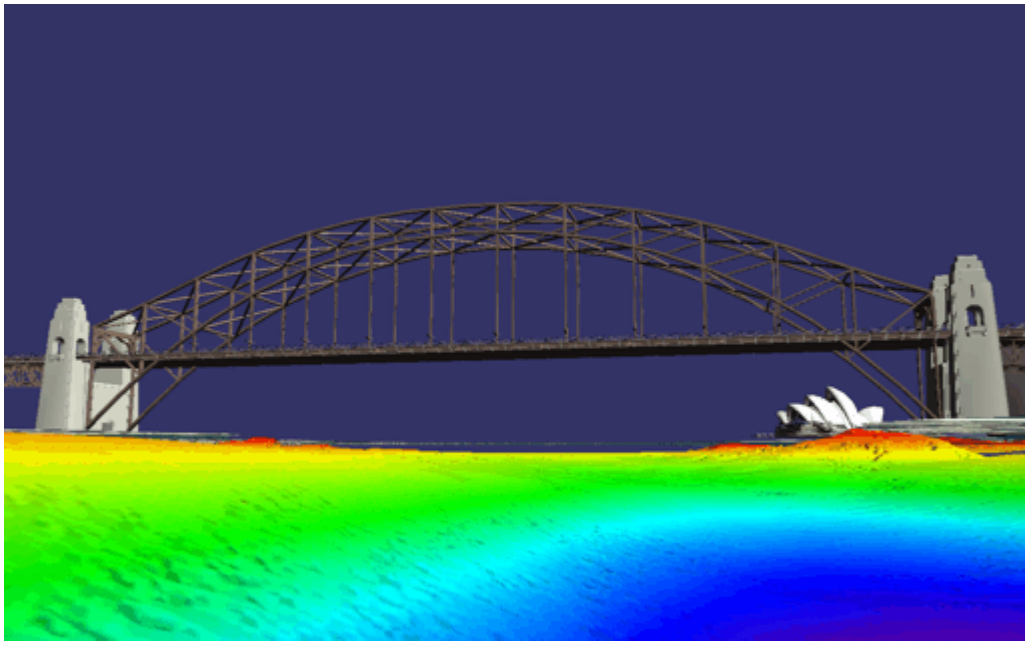
The file size of the image is shown in the dialog box. If you select a large dpi setting, the image is resampled to that setting.

6. Click **Save**.

The image is saved to the selected directory.

3-D Display Window

The 3-D Graphics window, configured in the 3-D Panel, shows project-level information in three dimensions. An example of the 3-D Graphics window is shown below (note that this display has been adjusted—see "SET 3-D WINDOW" ON PAGE 49).



In the 3-D Display window, georeferenced imagery is projected as an image layer (or layers) onto a height source. The BASE Surface is used as the terrain (or height source).

The layers that are visible in the Display window are shown in the 3-D display. The following layers can be viewed in the 3-D Display window:

- raster layers (BASE Surfaces, GeoTIFF images, BSB charts, etc.)
- ship track line vectors
- solid models

The 3-D Display is enhanced with a "fly-through" effect that makes it seem like you are gliding through the display. Movement is controlled by the mouse buttons. As you become more comfortable with manipulating the mouse buttons you will be able to apply seamless panning and zooming operations in this window.

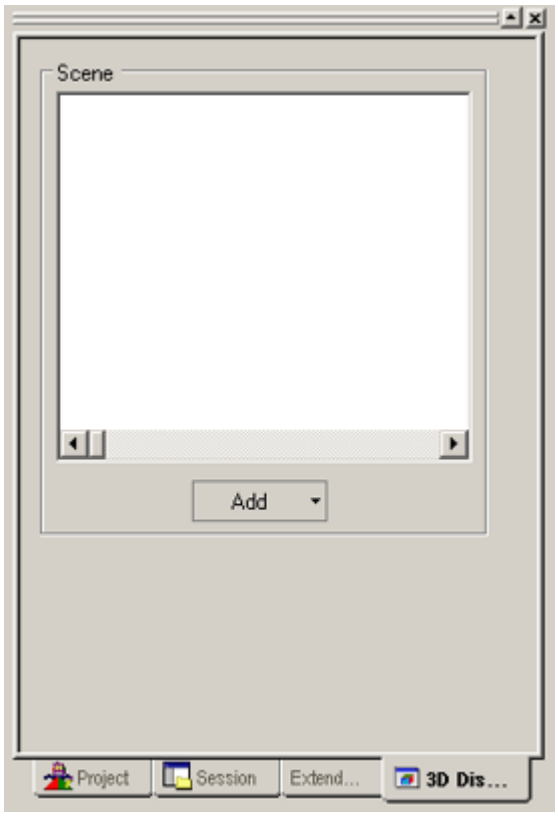
Set 3-D Window

View a 3-D representation of the dataset in the Display window by setting parameters on the 3-D Window tab in the Control window.

1. Adjust the Display window by turning layers on or off.
2. Select the command.

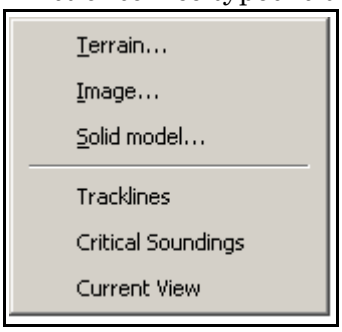
Menu	Window > 3-D Graphics Window
Tool	

The 3-D Graphics window is now visible in the Control window. This tabbed window provides the scene options for a 3-D view.



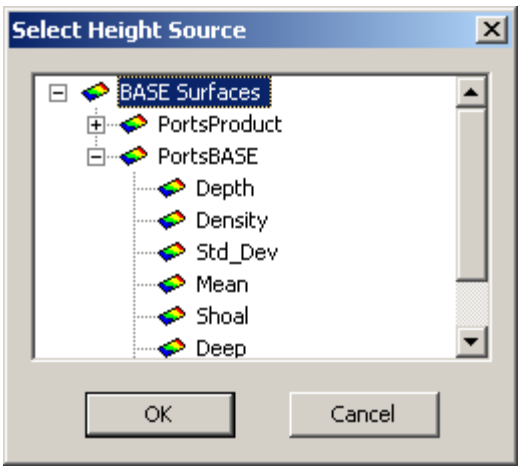
3. To add a scene, click **Add**.

A list of source types is displayed.



4. Select the Terrain option. You must select the Terrain option prior to selecting any other options in the list.

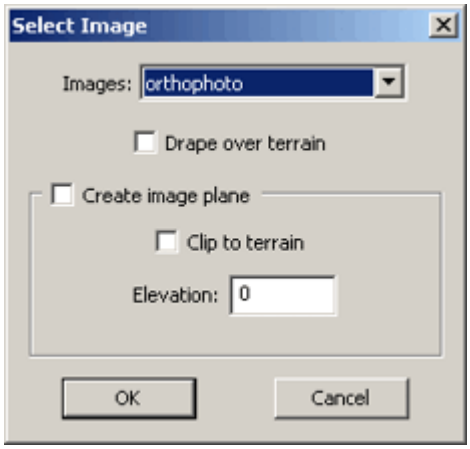
The following dialog box is displayed.



5. Select a terrain from a layer.
6. [Optional] Choose a source option:
   - If you select Image, go to step 8;
   - If you select Solid Model, go to step 14; or,
   - if you select Track Lines, go to step 16.

Image

The Select Image dialog box is displayed.



Select Image dialog box.

7. Select an image from the drop-down list. The image must already be opened in the Display window.
8. Select *Drape over terrain* to cover the selected terrain.
9. Select *Create image plane* to create a flat plane where there is no terrain.
10. Select *Clip to terrain* to constrain the image extents to that of the extents of the terrain area.
11. Type the *Elevation* to place the image on the same axis as the surface.
12. Click **OK**.

Solid Model

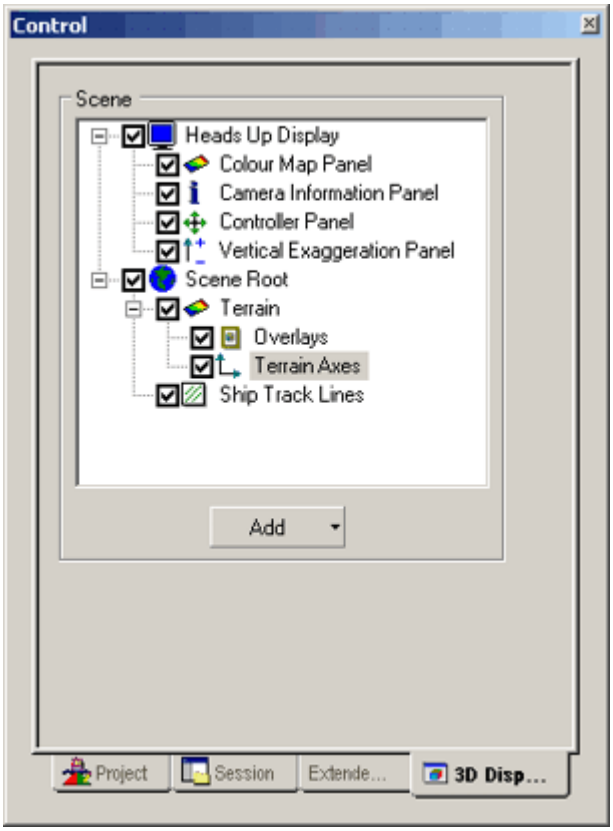
The standard Open dialog box is displayed.

13. Select a 3-D Studio Max File file to open.

The 3-D Graphics window display is updated with a scene attributes tree.

Track Lines

14. Select the Track Lines option to view track lines in the 3-D display.



You may choose to show or hide each layer or attribute by selecting or de-selecting the Scene options.

3-D Display Properties

You may change the properties settings, displayed in the Properties tab, for each attribute according to its type.

When selected, properties may be set for the following:

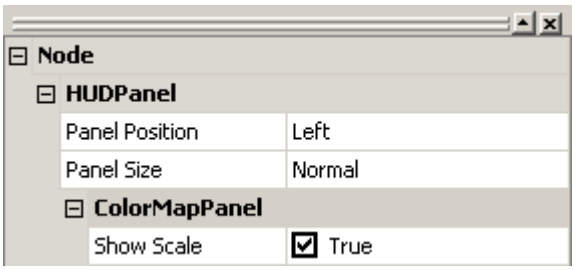
- Heads Up Display (HUD)
  - Colour Map Panel
  - Camera Information Panel
  - Controller Panel
  - Vertical Exaggeration Panel
- Scene Root
  - Terrain
    - Terrain Axes

After you change an attribute, update the display by clicking the mouse on the scene in the Display window.

Colour Map Panel

1. Select the Colour Map Panel attribute.

The Colour Map Panel properties are displayed (Properties tab).

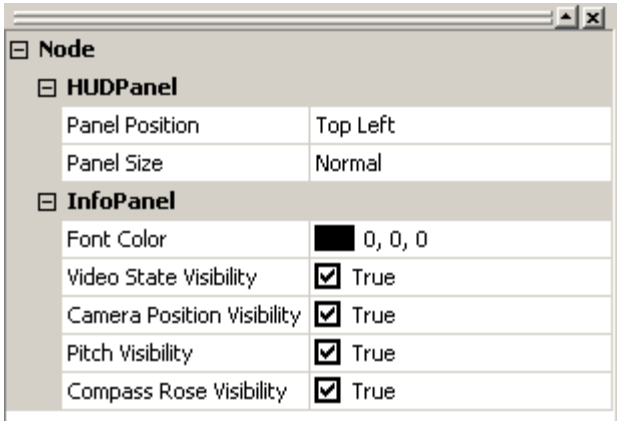


2. Select the desired settings for the following properties:
   - Panel Position: the position of the colour map in the Display window
   - Panel Size: the size of the colour map in the Display window
   - Show Scale: display the depth values for the colour map
3. Select the properties settings for the Colour Map Panel.

Camera Information Panel

1. Select the Camera Information Panel

The Camera Information Panel properties are displayed (Properties tab).

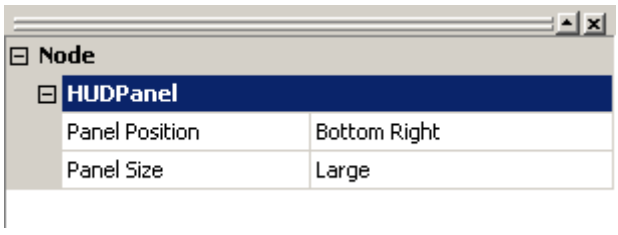


2. Select the settings for the following properties:
   - Panel Position: the position of the Camera Information Panel in the Display window
   - Panel Size: the size of the Camera Information Panel in the Display window
   - Font Colour: select colour for the font in the Camera Position window of the Camera Information panel (default is black)
   - Video State Visibility: select whether to have the video state portion of the Information panel on or off
   - Camera Position Visibility: select whether to have the camera position indicator on or off
   - Pitch Visibility: select whether to have the pitch indicator on or off
   - Compass Rose Visibility: select whether to have the compass rose on or off.

Controller Panel

1. Select the Controller Panel attribute.

The Controller Panel properties are displayed (Properties tab).

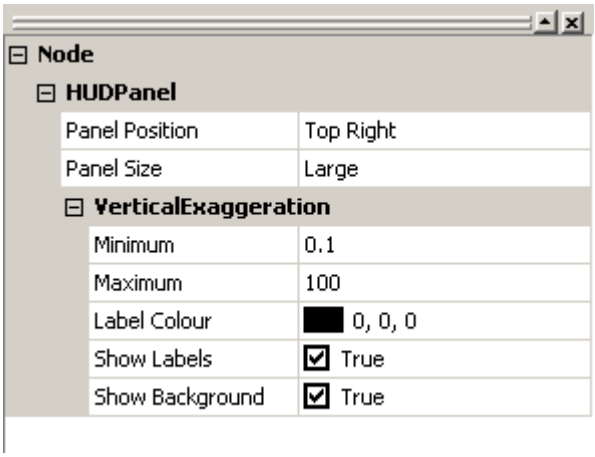


2. Select the desired settings for the following properties:
   - Panel Position: the position of the controller in the Display window
   - Panel Size: the size of the controller in the Display window

Vertical Exaggeration

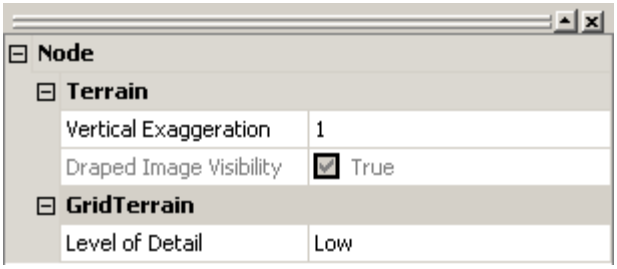
1. Select the Vertical Exaggeration Panel

The Vertical Exaggeration properties are displayed (Properties tab).



2. Set the following properties for the vertical exaggeration slider:
   - Set a *Minimum* and *Maximum* scale for the vertical exaggeration.
   - Select a *Label Colour* from the colour picker or create a custom colour from the palette.
   - Show or hide labels or the slider background by selecting or clearing the *Show Labels* or *Show Background* check boxes, respectively.
3. Select the Terrain attribute.

The Terrain properties are displayed (Properties tab).

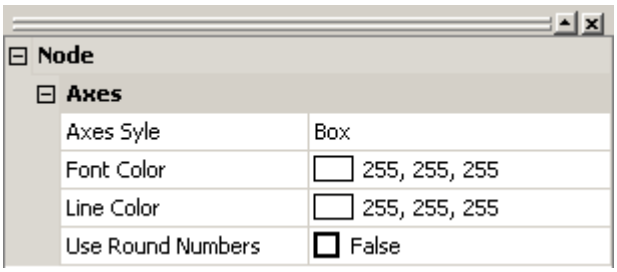


4. Select the settings for the following properties:
   - Vertical Exaggeration (Terrain): the amount of vertical, or shoal and depth, exaggeration to apply to the terrain display
   - Draped Image Visibility (Terrain): display an image file (e.g., GeoTIFF) relative to the selected surface
   - Level of Detail (Grid Terrain): determines how much detail is displayed as you zoom into the surface. A high value means a highly detailed surface is displayed. As you move out from the surface, the display is down-sampled so less detail is shown.

Terrain Axes

1. Select the Terrain Axes attribute.

The Terrain Axes properties are displayed (Properties tab).

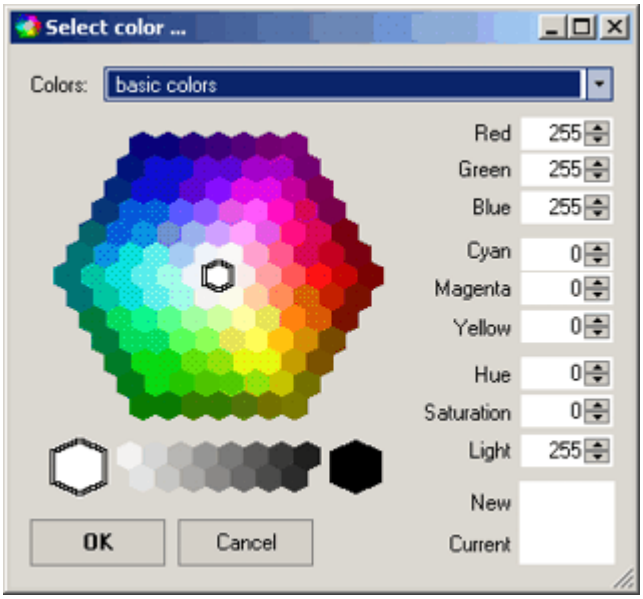


2. Select the desired settings for the following properties:
   - Axes Style: choose display option for the axes (grid lines)
   - Font Colour: colour for the displayed labels
   - Line Colour: colour for the grid plane lines

For the selected Colour option, you may type in RGB values in the field or select a pre-defined colour value from the drop-down list. To type in a colour value, go to step 22, or to select a colour value from the list, go to step 16.

3. Select a pre-defined colour.
4. [Optional] To create a colour, click **Custom Colour**.

The colour palette tool is displayed.



5. Select a colour picking scheme from the *Colours* list.
6. In the Custom Colour dialog box, the colour tool is on the left-hand side and the colour fields are on the right-hand side.
7. Select a colour or a setting from the colour tool;

   OR,

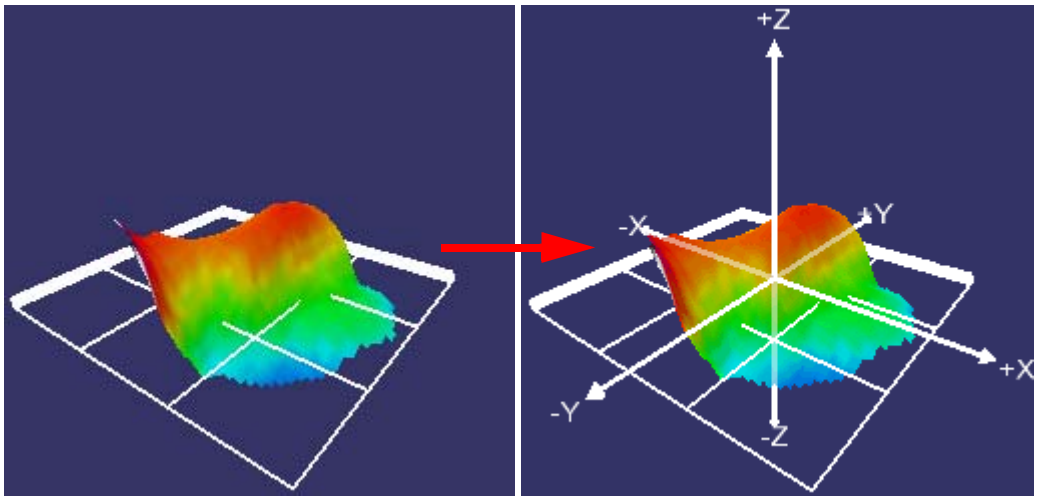
   type values into the appropriate colour fields.
8. To finish, click **OK**.

- Use Round Numbers: select this option to have all values displayed as whole numbers (e.g., 0.8 is displayed as 1, the nearest whole number).

To focus the view on a specific location of the surface, double-click your mouse button at the selected location.

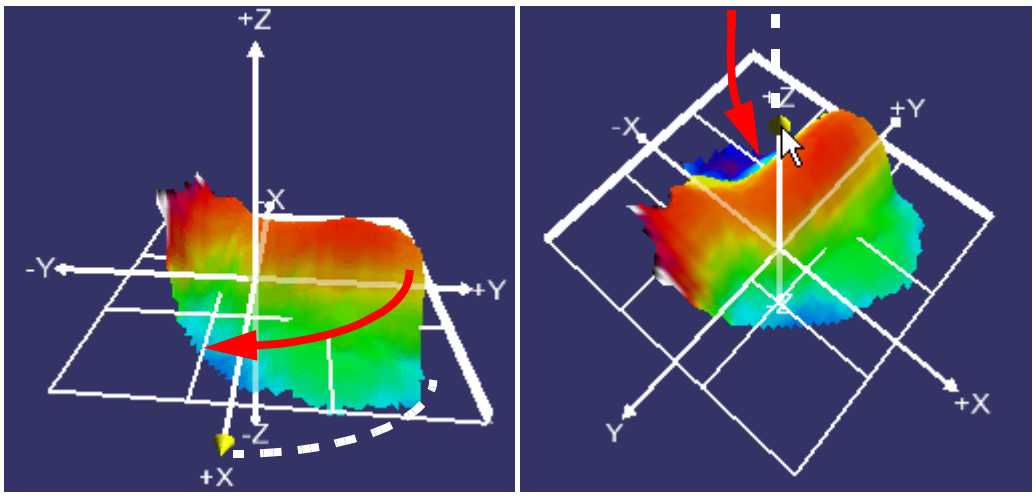
Control Axes

The control axes are displayed as an overlay to the surface on the grid. These controls allow you to rotate the view along the X, Y, or Z axes



1. To show or hide the control axes, click the Axis button on the 3-D toobar.

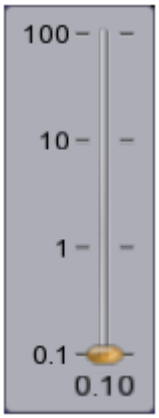
2. Select the end points of any of the axes to manipulate the view.
   - To rotate the view of the surface horizontally, click and hold the left mouse button on the end of the X or Y axis and move the mouse to the left or right of the Z axis in the display.
   - To rotate the view of the surface vertically, click and hold the left mouse button on the end of the Z axis and move the mouse up or down in the display.



Vertical Exaggeration

Vertical exaggeration is the ratio of the vertical scale to the horizontal scale. Increasing the vertical exaggeration can improve the display of features on the surface.

Controlling the vertical exaggeration is done using the vertical exaggeration scale slider. The slider is automatically displayed when the 3-D window is activated.



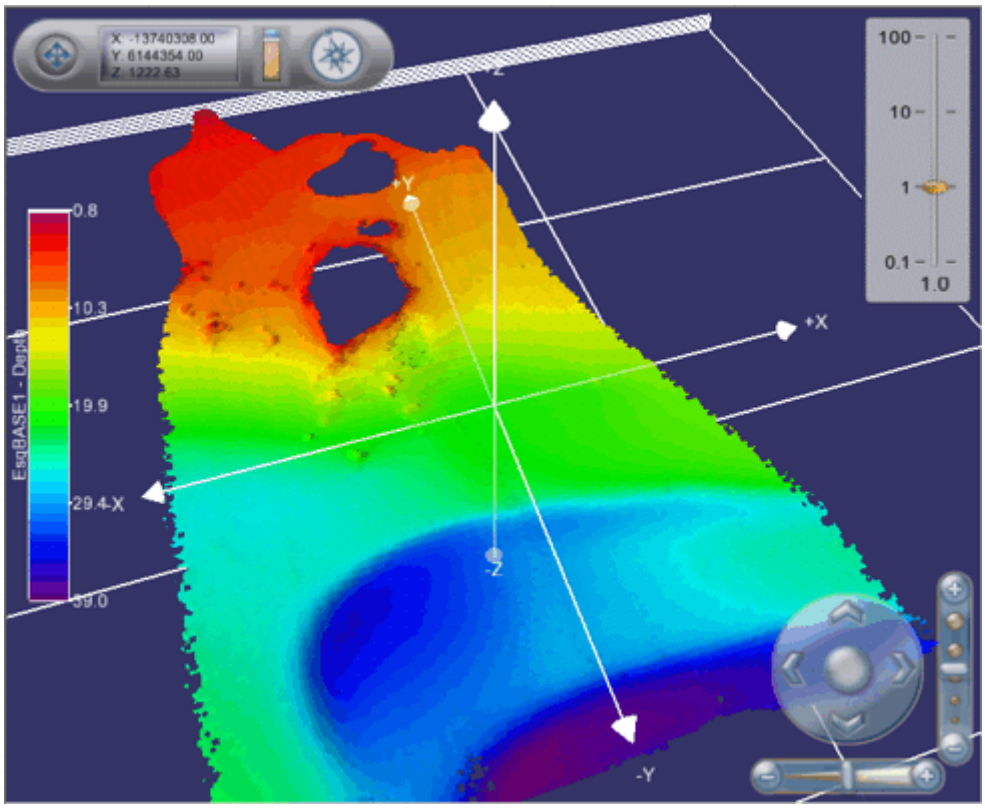
To use the slider:

1. Position the cursor over the button in the slider.
2. Press and hold the mouse to move button up or down the slider. As the button is moved the vertical exaggeration in the 3-D Display window is changed. The vertical exaggeration value is shown at the bottom of the slider.

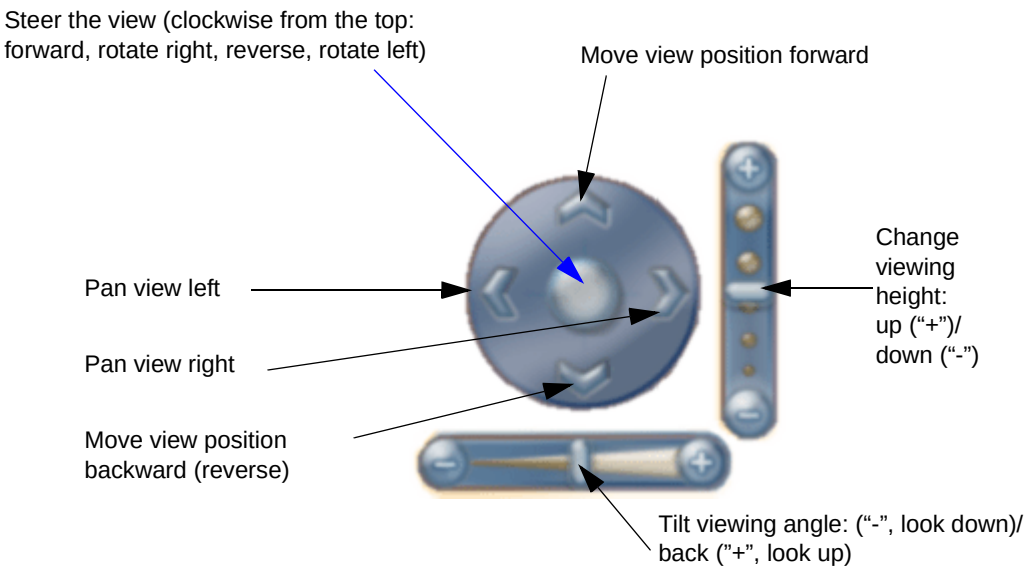
Navigate 3-D Display

In the 3-D Display window, you can zoom and pan to create a flying effect.

The flying effect is accomplished by using the Controller, the mouse (pressing and holding one or both of the mouse buttons while simultaneously moving the mouse pointer in the direction you want to move) or the keyboard keys.



You may use the Controller directional pad and slider controls to modify the view. The controls for the Controller are as follows:


Steer the view (clockwise from the top: forward, rotate right, reverse, rotate left)
Move view position forward
Pan view left
Pan view right
Move view position backward (reverse)
Change viewing height: up (“+”)/ down (“-”)
Tilt viewing angle: (“-”, look down)/ back (”+”, look up)

When using the mouse pointer to navigate the speed you travel through the display or pan the view varies with the pointer's distance from the centre (stationary) point.

The pointer position in the display indicates the direction you will move. The mouse button determines what operations are available to you.

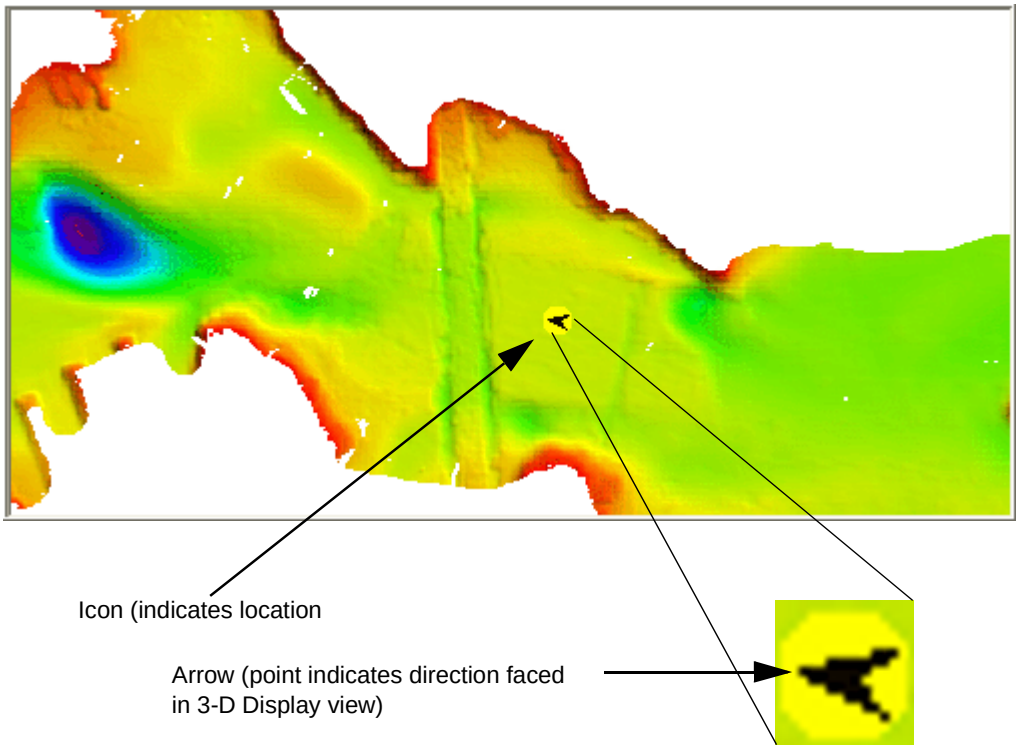
You may also use the navigation icon displayed in the 2-D Display window of the selected surface.

Camera Information Panel

As you zoom and pan, the Camera Information Panel reflects the position and orientation of the camera. The panel is comprised of various indicators, as follows:

2-D Display icon

The display icon in the Display window shows the location and view direction you are currently at in the 3-D window.



You can use this icon to navigate through the 3-D Display window by “grabbing” the icon (pressing and holding the mouse button) and moving it in the 2-D Display. The display in the 3-D window “flies” to the location where the icon is positioned. The direction of the arrow indicates the compass direction you are facing.

The following table lists the available actions.

Cursor position	Mouse Button	Key	Action (view)
Centre	—	—	None.
Up	Left	W	Pan forward in a straight line (horizontal to grid).
	Right	↑	Tilt back (viewing angle goes up).
	Left+Right	R	Pan up in a straight line (vertical to grid).
Down	Right	S	Pan backward in a straight line (horizontal to grid).
	Left	↓	Tilt forward (viewing angle goes down).
	Left+Right	F	Pan down in a straight line (vertical to grid).
Left	Left+Right	A	Pan left in a straight line (horizontal to grid).
Right	Left+Right	D	Pan right in a straight line (horizontal to grid).
Left	Left	←	Rotate left.
Right	Left	→	Rotate right.
—	Scroll forward	C	Zoom in
—	Scroll reverse	V	Zoom out
—	Double-click	—	Zoom into the double-clicked point on the display

1. Use the cursor position and mouse buttons, keyboard keys or Controller Panel to navigate (pan and zoom) through the display in the 3-D Display window.

3-D Display Fly-through options

You may make and save movies of a 3-D Display scene fly-through for playback at a later time. The controls for creation and playback of your movies are located in the 3-D Display toolbar.

Record a fly-through

1. With a scene open in the 3-D Panel and shown in the Display window, select a starting point for your fly-through.
2. On the 3-D Display toolbar, click the ● (Record Flight) button.
3. Use the controls to navigate through the 3-D Display window.

4. To stop recording your fly-though, click the Record Flight button again.

Your fly-through is ready for playback.

Play the fly-through

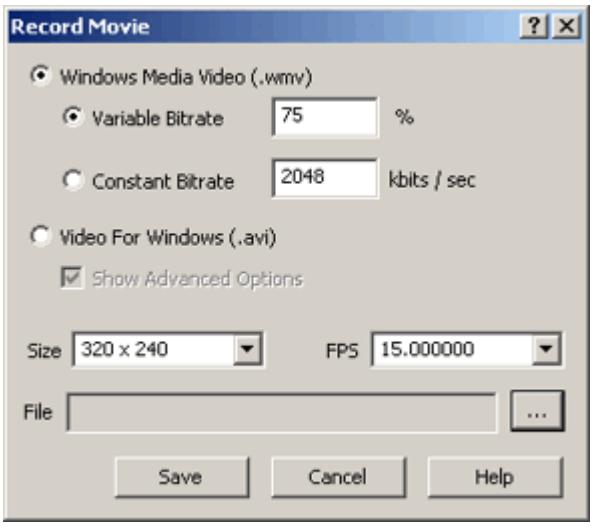
1. Click the ▶ (Play Flight) button.
2. [Optional] To pause the playback, click the ⏸ (Pause Flight) button.

The flight playback stops after the recording is finished playing.

Record a movie

1. Click the (Record Movie) button.

The Record Movie dialog box is displayed.



2. Choose the format of the video file.
   - If you select AVI format, go to step 5; or,
   - if you select WMV format, go to step 3.

You may select a bit rate for the recorded data as a variable (percentage of the total) or a constant (kilobits per second) value.

3. Select a bit rate option.
4. Type a value for the selected bit rate and go to step 6.
5. [Optional] To see the advanced playback options for your AVI movie before it is saved, select the *Show Advanced Options* option.

You must select image size and FPS (Frames-Per-Second) options for the playback of your movie.

6. Select a Size option.
7. Select a FPS option.
8. To select the save name and location for your movie file, click the Browse button (**...**).

A Save As dialog box is displayed.

9. Type a save name.
10. Select a save folder.

11. Click **Save**.

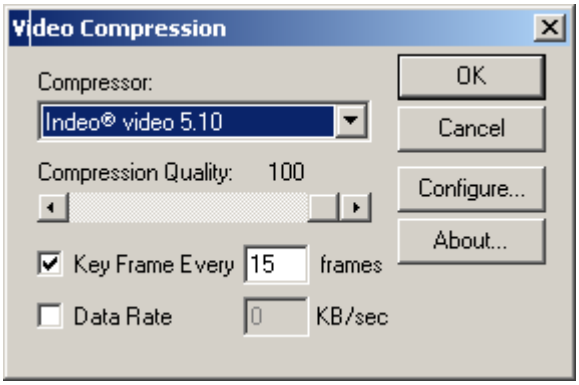
The Save As dialog box is closed, returning you to the Record Movie dialog box.

12. If you clicked the WMV format option, click **Save** and go to step 19;

    OR,

    if you clicked the AVI format option, click **Save** and go to step 13.

The Video Compression dialog box is displayed.



13. Select a *Compressor*.

> Some video compressors do not allow you to set the following options (in steps 14 to 17) for your movie files.

14. Use the slider control to select the compression quality (displayed as a value in the range of 0 and 100 percent).
15. [Optional] To insert key frames, select the *Key Frame* option and type a *frames* value.
16. [Optional] To set the data rate, select the *Data Rate* option and type a *KB/sec* (kilobits per second) value.

With some compressors, you may also choose the configuration settings to use for your AVI movie.

17. [Optional] To configure the selected compressor, click **Configure** (select from the available configuration options).
18. Click **OK**.

*A file saved message is displayed.

19. To finish, click **OK**.

The process is finished.

Cut, Copy and Paste

The Cut, Copy and Paste commands are useful for manipulating files in the Control window. These commands are similar to those used in other Windows applications.

Line folders can only be moved or copied between Day folders and Day folders can only be moved or copied between Vessel folders.

Cut

The Cut command moves selected Vessel, Day, or Line folders to the clipboard.

1. Select the object you want to cut.
2. Select the Cut command.

The file names in the Control window are dimmed.

3. You can undo the Cut command by pressing the <Esc> key.

Menu	Edit > Cut
Tool	
Pop-up	Cut
Key	<Ctrl+X>

Copy

The Copy command reproduces the selected object without removing it from the Control window.

1. Click the object you want to copy.
2. Select the Copy command.

Menu	Edit > Copy
Tool	
Pop-up	Copy
Key	<Ctrl+C>

Select All and Copy (Worksheet window)

Select and copy data in the *Query* tab of the Information to the clipboard. The data can then be placed in another application.

1. Click inside the Worksheet window and select the Select All command.
2. Select the Copy command.

The data is copied to the Windows clipboard.

Paste

The Paste command takes an object that has been cut or copied and places it in another location.

1. Make sure you have cut or copied an object.
2. Place the cursor at the location you want to place the object.
3. Select the Paste command.

The object is pasted into the application at the selected location.

Menu	Edit > Paste
Tool	
Pop-up	Paste
Key	<Ctrl+V>

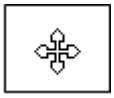
Measure Distance

Menu	Tools > Measure Distance
Tool	

Measure the distance between two points in a survey area or in the 2-D Subset View. The distance is shown according to the units you selected in the Display tab of the Options dialog box (see "UNITS" ON PAGE 95).

1. Select the Measure Distance command.

The Measure Distance cursor icon is displayed.



2. Press and hold the mouse button and drag the cursor along the Display window.

As the cursor is dragged across the window, a line is drawn from the start point to the point where the mouse button is released. The distance between the two points is shown along the line.

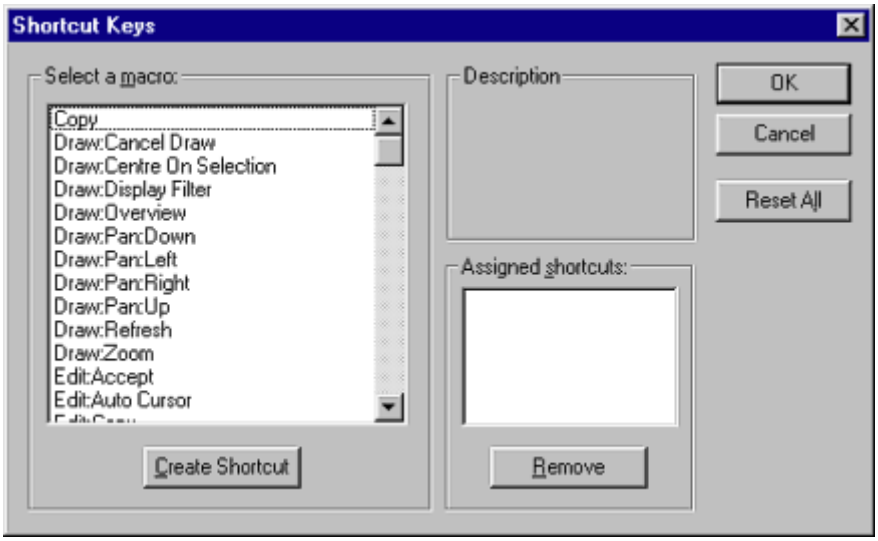
Keyboard Shortcuts

You can customize existing shortcuts, or create new shortcuts for menu commands that do not possess one. The keyboard shortcut definitions are stored, by default, in …\\Hips\bin\Hips.mac.

1. Select the Shortcut Keys command.

Menu	Tools > Shortcut Keys

The Shortcut Keys dialog box is displayed.



2. Click a menu command in the *Select a Macro* list box.
   - If the command already has a shortcut, the keys are displayed in the *Assigned Shortcuts* box. A brief explanation of the shortcut is displayed in *Description* field.
   - If the command does not have a shortcut then these two sections are empty.

Modify existing shortcuts

3. To delete an existing shortcut, highlight the shortcut in the *Assigned Shortcuts* area and click **Remove**.

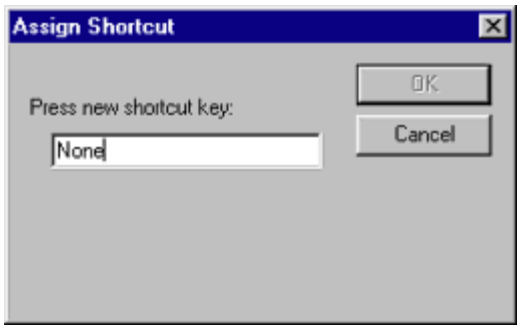
The shortcut is no longer active in the project.

Create a new shortcut

You can create new shortcuts to replace current shortcuts or create shortcuts for commands without one.

4. Click **Create Shortcut**.

The Assign Shortcut dialog box is displayed.



You can create a shortcut using function keys, control keys or regular keys. You can also combine keys.

5. Type a new shortcut in the text area. As you type the keys are displayed in the text area. Already assigned shortcuts are displayed below the text box.
6. Once you have selected the key(s), click **OK** to close the dialog box.

The shortcut is displayed in the Shortcut Keys dialog box.

7. If you want to restore previous shortcuts, click **Reset All**.

The new shortcut is deleted and the previous default command is re-assigned.

8. Click **OK** to save the settings and close the dialog box.

Save, Close and Exit

Save Projects

Save changes to data after cleaning or when the display in a session file is changed.

1. Select the Save command.

Menu	File > Save
Tool	
Key	<Ctrl + S>

You can automatically save data-cleaning changes at regular intervals by selecting the Auto Save option (see "GENERAL" ON PAGE 75).

Close a Project

1. Select the Close Project command, or click the Windows close icon on the top right of the window.

You are prompted to save any changes.

2. Click **OK** to save changes.

Menu	File > Close Project
Tool	
Key	<Ctrl > F4>

Close individual track lines

1. Select a track line file in the Control window, or select a track line in the Display window.
2. Right-click the mouse and select a Close Lines command.

The track line closes.

Pop-up	Close Lines

Exit HIPS and SIPS

1. Choose the Exit command.
2. You are prompted to save any changes, if necessary.
3. Click **OK**.

HIPS and SIPS is closed.

Menu	File > Exit
Key	<Alt + F4>

3

Options

Introduction

The Options dialog box has a number of parameters to help you control the appearance of the interface, the behaviour of some commands, and the system environment.

1. Select the Options command.

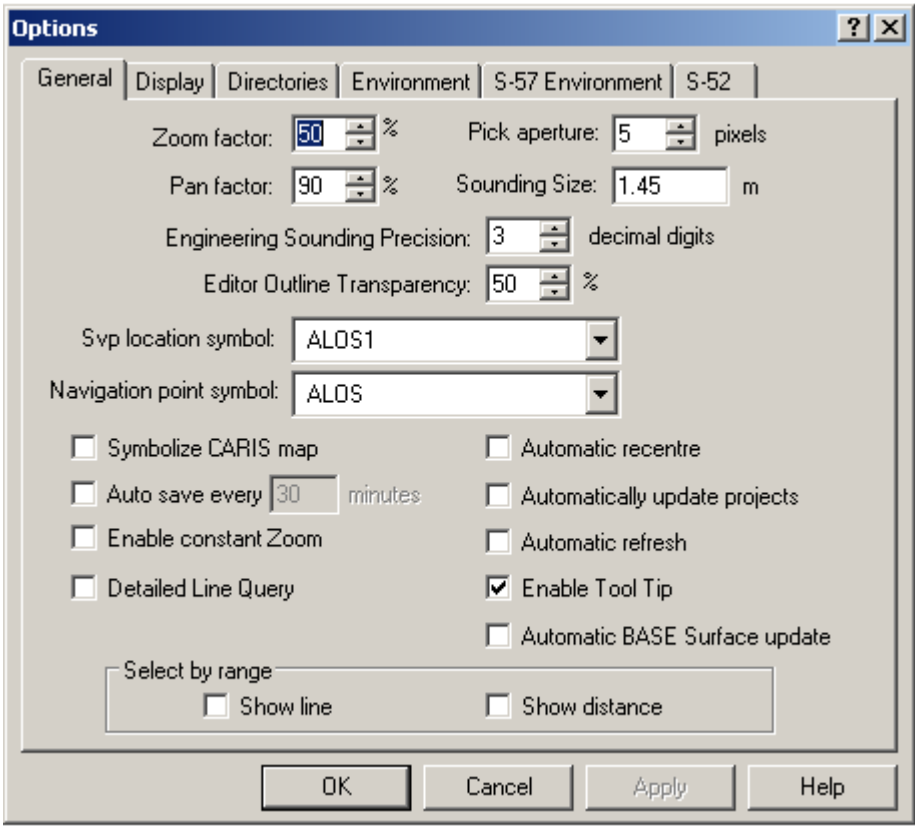
Menu	Tools > Options

The Options dialog box is displayed. The dialog box contains the following five tabs:

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

General

On the Options - General tab you can set parameters that determine how you want to work with data.



1. Set the parameters described in the table below.

Field	Description	To assign a value...
Zoom Factor	Set the percentage to increase or decrease the display scale when using the Zoom functions.	Select a percentage rate for the zoom tool by clicking the up or down arrows beside the *Zoom Factor* box.
Pan Factor	Set the percentage used to move up, down, left, or right while using the Pan tools.	Select a percentage by clicking the up or down arrow keys beside the *Pan Factor* box.
Pick Aperture	The Pick Aperture determines the maximum distance (in pixels) that a cursor must be from an object to select it. For example, if the Pick Aperture is set to five, the cursor must be a distance of five pixels (or less) to select an object.	Select the pixel area for the Pick Aperture by clicking the up and down arrow buttons.
Sounding Size	Display sounding digits in a field sheet or tiled area at a selected size (millimetres at map scale).	Enter a sounding size value in the *Sounding Size* box.

Field	Description	To assign a value...
Engineer Sounding Precision	Set the number of places to the right of the decimal point that digits are shown for engineering sounding display.	Select the precision value by clicking the up or down arrow buttons.
Editor Outline Transparency	Control the transparency levels of fills for Swath, Side Scan and Subset Editor outlines in the Display window.	Select a percentage value for transparency (0 equals no transparency, 100 equals total transparency) by clicking the up or down arrow buttons.
SVP Location Symbol	Select a feature code for the symbol that shows the location where SVP data was recorded.	Select a feature code from the drop-down list.
Navigation Point Symbol	Select a feature code for the symbol display of points in the Navigation Editor.	Select a feature code from the drop-down list.
Symbolize CARIS Map	This option draws the background CARIS map with its associated line patterns and symbolization.	Make sure the *Symbolize CARIS Map* box is checked to implement this option.
Auto Save	Automatically save all status flag changes at set intervals.	Select the *Auto Save* check box to implement this function. Enter the interval (in minutes) that work is to be saved.
Enable Constant Zoom	Control whether the Zoom function allows repeated Zoom commands until the Zoom function is turned off.	Make sure the *Enable Constant Zoom* box is checked to implement this option.
Detailed Line Query	Display all data fields for a track line when it is queried.	Select the *Detailed Line Query* command.
Automatic Recentre	Redraw the Display window so that the sounding coverage outline is always displayed in the centre when scrolling along the line.	Select the *Automatic Recentre* check box.
Automatically Update Projects	Save the UTM zone setting in a project so HIPS/SIPS does not have to search for the zone when a project is reopened. This option is only available with Auto UTM.	Make sure the *Automatically Update Projects* box is checked to implement this option.
Automatic Refresh	Refresh the Display window automatically when layers are turned on or off, or when the drawing order is changed in the Control window.	Make sure the *Automatic Refresh* box is checked to implement this option.
Enable Tool Tip	Show/hide the display of ToolTips when the cursor is over a BASE surface or toolbar button.	Make sure the *Show Tool Tip* box is checked to implement this option.
Automatic BASE Surface Update	Automatically update BASE Surfaces after cleaning sounding data in the Swath and Subset Editors, or after cleaning CUBE data in the Subset Editor.	Select or clear the *Automatic BASE Surface Update* check box.

Field	Description	To assign a value...
Show Line	Display a diagonal line inside the bounding box that is drawn when the mouse button is pressed down and the cursor is dragged across the screen.	Click the *Show Line* check box.
Show Distance	Display the distance that the bounding box covers when it is drawn on the screen.	Click the *Show Distance* check box.

2. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Display

Menu	Tools > Options

1. Select the Options command.

The Options - Display dialog box is displayed.

2. Select the Display tab.

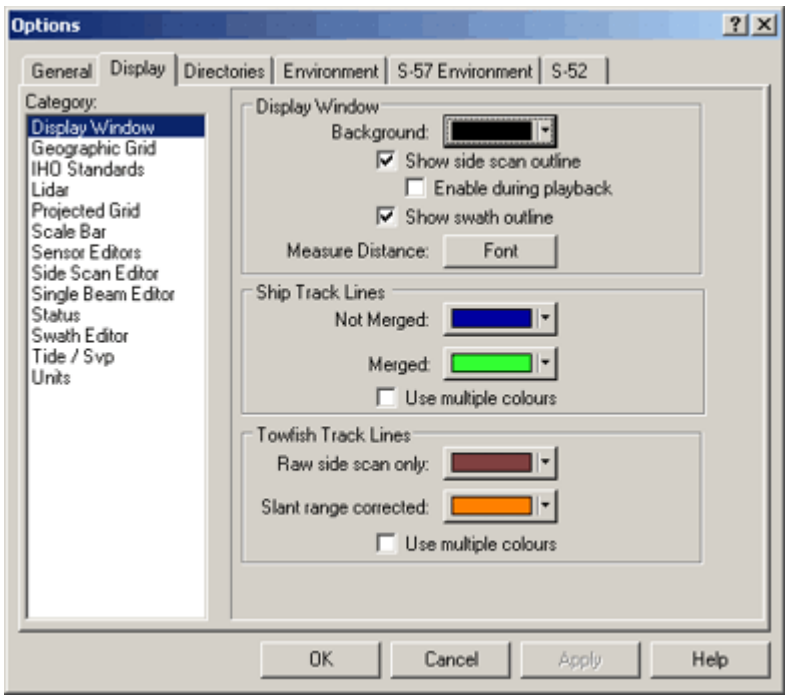
The Display tab has a list of various display controls.

The controls for the selected editor or window are displayed on the right side of the window. Refer to the appropriate section below for instructions to set the parameters for each window or editor.

Display window

The Tools >Options - Display Window tab controls the display and colour features for the ship and towfish track lines.

1. Select Display Window from the *Category* list.



2. Set the parameters described in the table below.

Field	Description	To assign a value...
Display Window fields:		

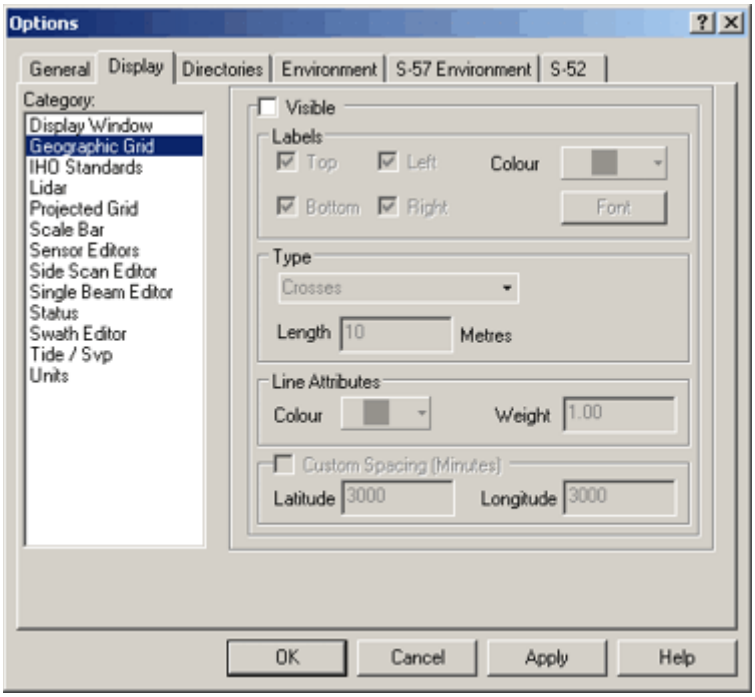
Field	Description	To assign a value...
Background	Control the background colour.	Select a colour for the background from the colour palette, or create a custom colour from the standard Windows colour picker.
Show Side Scan Outline	The side scan outline marks the extent of the data currently visible in the Side Scan Editor.	Select the *Show Side Scan Outline* check box to display the marker in the Display window while the Side Scan Editor is open.
Enable during playback	Move the side scan outline along the track line during the Playback option in the Side Scan Editor.	Select the *Enable During Playback* check box to make this option active.
Show Swath Outline	The swath outline marks the extent of the data currently visible in the Swath Editor.	Select the *Show Swath Outline* check box to display the marker in the Display window while the Swath Editor is open.
Measure Distance	Select a font to display measurements when the Measure Distance command is used.	Click **Font** and select a type face, font size and colour from the dialog box.
Ship Track Lines fields		
Not Merged	The colour for track lines that have not been merged into a position/depth file	Select a colour from the colour palette, or create a custom colour from the standard Windows colour picker.
Merged	The colour of track lines that have been merged into a position/depth file.	
Towfish Track Lines fields		
Raw Side Scan Only	The colour used to display track lines when side scan data is still in raw mode.	
Slant Range Corrected	The colour used to display track lines when side scan data has been slant range corrected.	
Use Multiple Colours	This option uses a set of ten distinct colours to colour the track lines in the Display window. This is helpful if the survey data was logged as multiple files along the same runline.	Click the *Use Multiple Colours* check box to enable this option for ship track lines or towfish lines.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Geographic Grid

The Options - Geographic Grids tab controls the options for displaying grids based on geographic units.

1. Select Geographic Grid from the *Category List*.



2. Set the parameters described in the table below.

Field	Description	To assign a value...
Visible	Show a grid based on geographic coordinates in the Display window.	Select the check box.
Labels: Top/Left/Bottom/ Right	Display geographic coordinates at selected locations in the Display window	Select the check boxes.
Colour	Set a colour for the grid labels.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Font	Typeface and point size for grid labels	Click the Font button to open the standard Windows Font dialog box to select a font type and size.
Type	The type of markings for the geographic grid.	Select a type from the drop-down list.
Length	The length of the markings (mm at map scale).	Type the length.

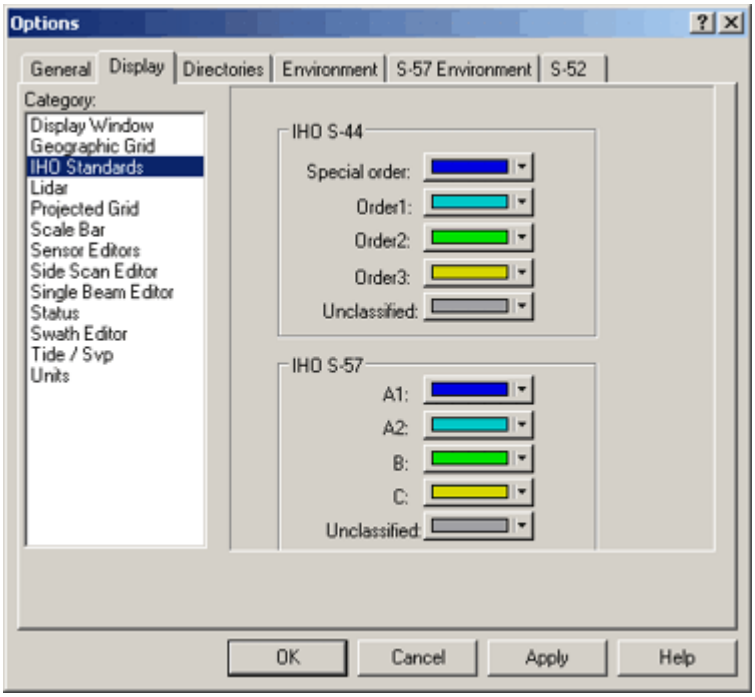
Field	Description	To assign a value...
Line Attribute Colour	Colour for the geographic grid markings.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Weight	Thickness of the markings	Type a value into the field.
Custom Spacing	Position the grid markings at selected locations in the workspace.	Select the check box.
Grid Spacing	Spacing of grid markers in mm at map scale.	Type a value in the provided field.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

IHO Standards

The Options - IHO Standards tab controls the colour options in the Subset Editor to display data according to International Hydrographic Organization (IHO) standards. Colours can be set for each S-44 survey order and S-57 Zone of Confidence (CATZOC). The *Rejected* option sets the colour for soundings that exceed defined limits of the selected standard

1. Select IHO Standards from the *Category* list.



2. Set the parameters described in the table below.

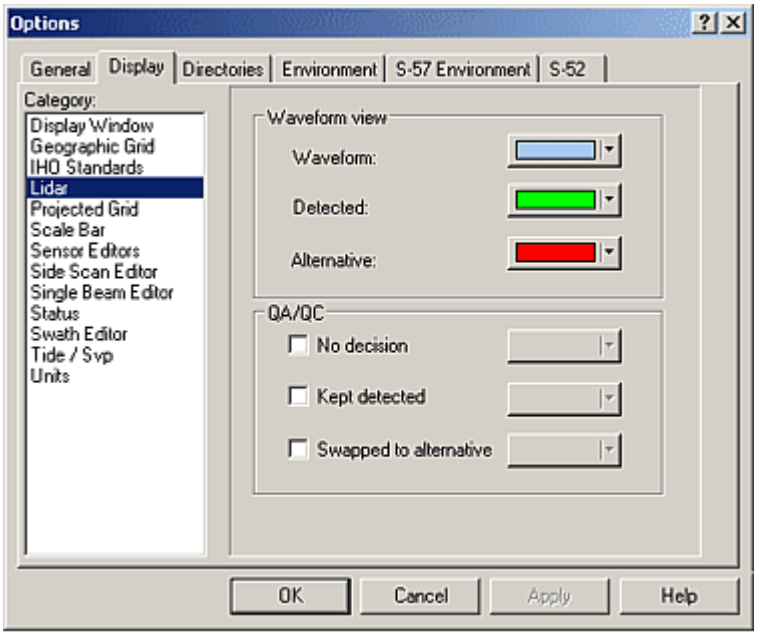
Field	Description	To assign a value...
IHO S-44	Colours for displaying soundings according to S-44 survey order.	Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker.
IHO S-57	Colours for displaying soundings according to the S-57 CATZOC zones of confidence attributes.	

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Lidar

The Tools> Options - Lidar tab controls the colour options for lidar data in the Swath and Subset Editors.

1. Select Lidar from the *Category* list.



2. Set the parameters described in the table below.

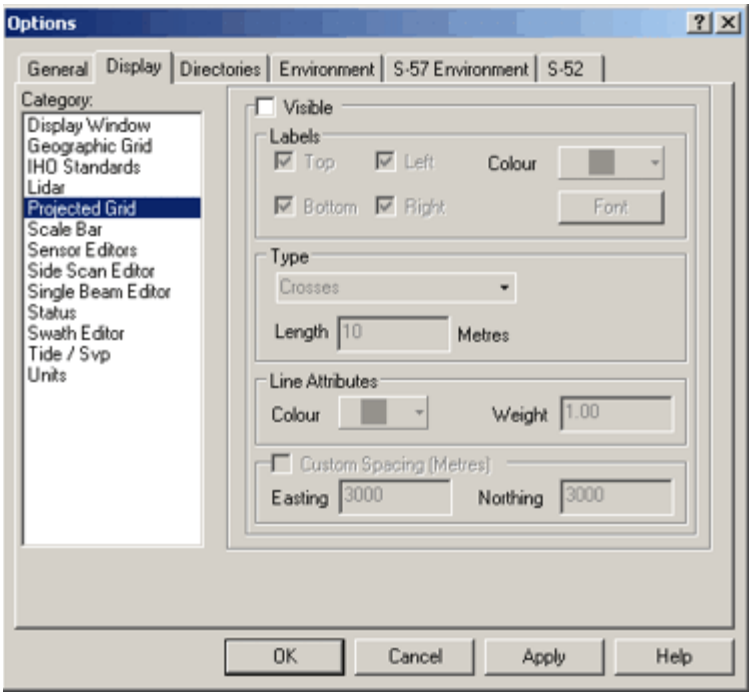
Field	Description	To assign a value...
Waveform view:		
Waveform	Selected colour for displaying waveform data.	Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker.
Detected	Selected colour for displaying soundings.	
Alternative	Selected colour for displaying soundings with an alternative depth.	
QA/QC:		
No Decision	Display soundings that have a detected and alternative depth (Quality 1 flag)	Display or hide soundings with specific quality flag by selecting or clearing any of the quality flag check boxes. Select a new colour for each of the lidar flags from the colour palette, or create a custom colour from the standard windows palette.
Kept Detected	Display soundings where the detected depth is retained over the alternative depth (Quality 2 flag)	
Swap to Alternative	Display soundings where the detected depth has been replaced by the alternative depth (Quality 3 flag)	

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Projected Grid

The Options - Projected Grid tab controls options for displaying a grid based on the selected projection.

1. Select Projected Grid from the Category list.



2. Set the parameters described in the table below.

Field	Description	To assign a value...
Visible	Show a grid based on geographic coordinates in the Display window.	Select the check box.
Top/Left/Bottom/ Right	Display geographic coordinates at selected locations in the Display window	Select the check boxes.
Colour	Colour for the grid labels.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Font	Typeface and point size for grid labels	Click the Font button to open the standard Windows Font dialog box to select a font type and size.
Type	The type of markings for the geographic grid.	Select a type from the drop-down list.
Length	The length of the markings (mm at map scale).	Type the length.

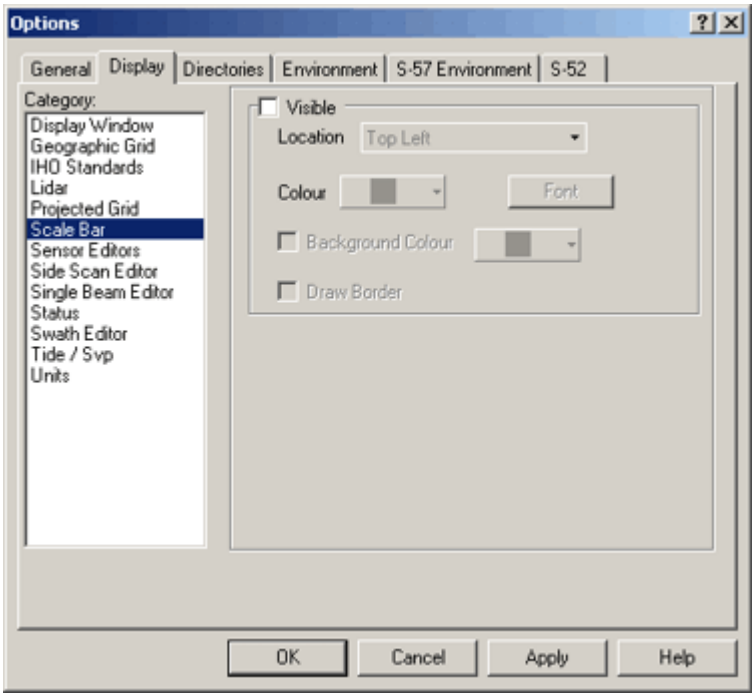
Field	Description	To assign a value...
Line Attribute Colour	Colour for the geographic grid markings.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Weight	Thickness of the markings	Type a value into the field.
Custom Spacing	Position the grid markings at selected locations in the workspace.	Select the check box.
Grid Spacing	Spacing of grid markers in mm at map scale.	Type a value in the provided field.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Scale Bar

The Options - Scale Bar tab controls the options for the scale bar.

1. Select Scale Bar in the Category list.



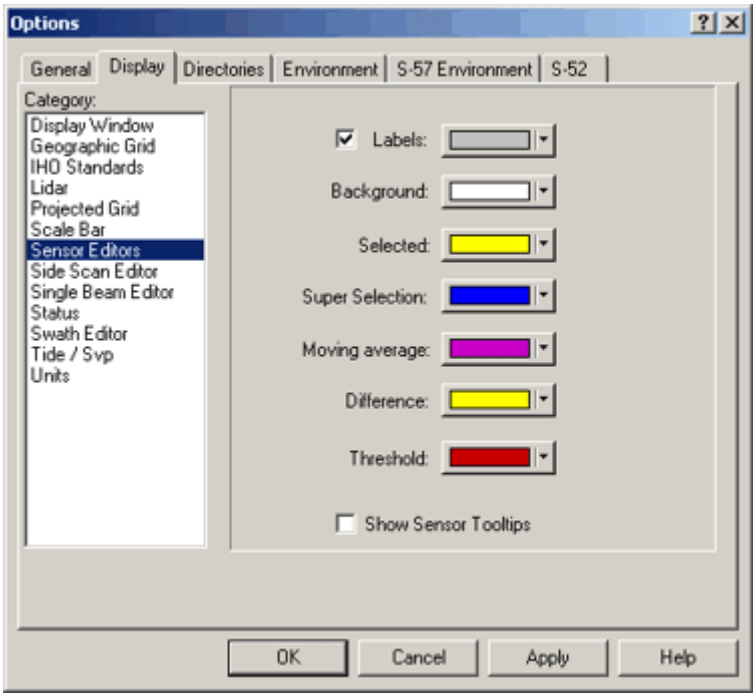
.

Field	Description	To assign a value...
Visible	Show a scale bar in the Display window.	Select the check box.
Location	Position of the scale bar in the window.	Select a location from the drop-down list.
Colour	Colour of the scale bar.	Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Font	Select the font type, style and size from the Font dialog box.	Click the Font button to open the standard Windows Font dialog box to select a font type and size.
Background Colour	Display a colour fill for the scale bar area.	Select the check box to implement this option. Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
Draw Border	Draw a border line around the scale bar area.	Select the check box to implement this option.

Sensor Editors

The Tools >Options - Sensor Editors tab controls some of the colour settings for the editors.

1. Select Sensor Editor from the *Category* list.



2. Set the parameters described in the table below.

Field	Description	To assign a value...
Labels	Labels display the horizontal and vertical scale lines and annotations in the editors.	Show or hide labels (the labels are toggled to on when the Label box is checked). Select a colour for the labels from the colour palette, or create a custom colour from the standard Windows colour selector.
Background	This feature controls the background colour in the editors.	Select a colour from the colour palette, or create a custom colour from the standard Windows colour selector.
Selected	The colour for highlighted data in the Display window and in the editors.	
Superselection	The colour for superselected lidar data in the Swath and Subset Editors.	

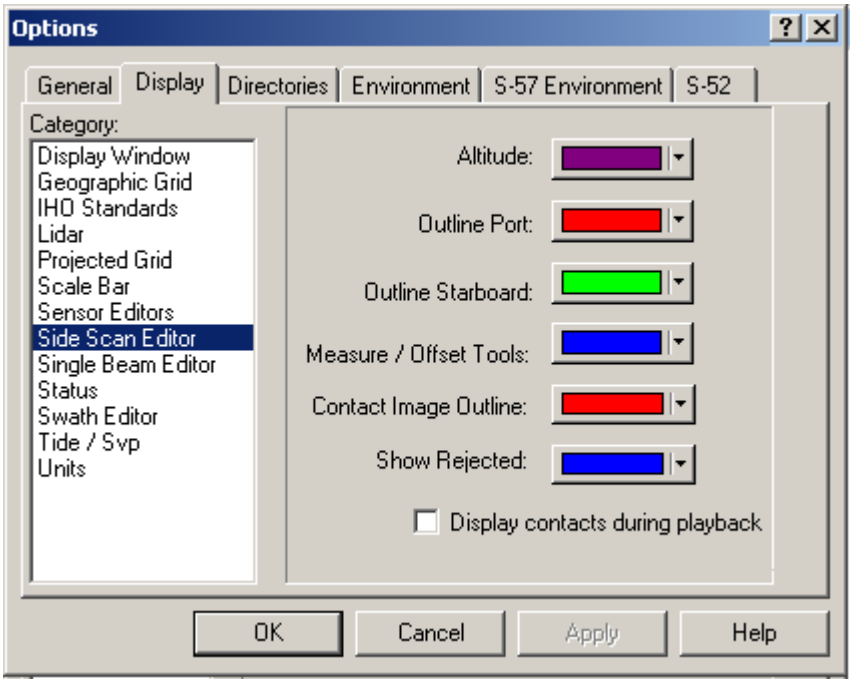
Field	Description	To assign a value...
Moving Average	The colour for the Moving Average line in the Attitude and Single Beam Editors.	Select a colour from the colour palette, or create a custom colour from the standard Windows colour selector.
Difference	The colour for Difference line in the Attitude and Single Beam Editors.	
Threshold	The colour for the Threshold line in the Attitude and Single Beam Editors.	
Show Sensor ToolTips	ToolTips are used in the Attitude and Navigation Editors for displaying data values as the cursor moves over the graphs.	Show or hide tools tips by clicking the *Tool Tips* check box (checked means this feature is visible).

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Side Scan Editor

The Tools >Options - Side Scan Editor Display tab controls the colours used for the Side Scan Editor.

1. Select Side Scan Editor from the *Category* list.



2. Set the parameters described in the table below.

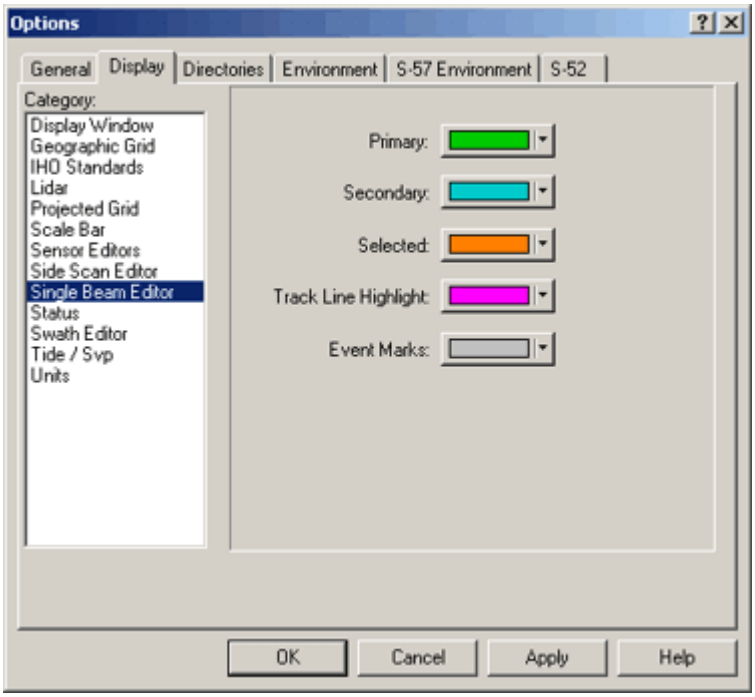
Field	Description	To assign a value...
Altitude	The colour of the digitized line along the seabed that is used determine the height of the towfish from nadir when slant-range correction is performed.	Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker.
Outline Port	The colour for the port side of the side scan coverage outline in the Display window.	
Outline Starboard	The colour for the starboard side of the coverage outline in the Display window.	
Measure /Offset Tools	The colour for displaying the values when using the Measure Distance and Measure Shadow tools, and the colour of the range display.	
Contact Image Outline	The colour for displaying the outline of a contact.	
Show Rejected	The colour of rejected pings. (visible when the Show Rejected button on the Toolbar is activated)	
Display contacts during playback	Make contacts visible in waterfall window during scrolling or playback.	Select the check box.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Single Beam Editor

The Options - Single Beam Editor tab controls the colours for the Single Beam Editor.

1. Select Single Beam Editor from the *Category* list.



2. Set the parameters described in the table below.

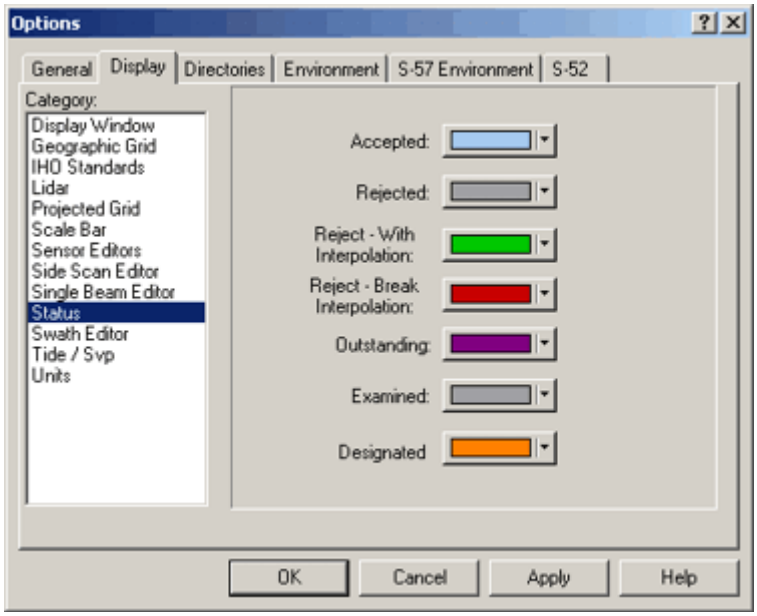
Field	Description	To assign a value...
Primary	Colour of primary frequency soundings in a dual-frequency system.	Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker.
Secondary	Colour for secondary frequency soundings in a dual-frequency system.	
Selected	Colour for soundings that are selected for further processing. Selected data begins as a copy of primary frequency data, but can be modified to include secondary frequency data.	
Track Line Highlight	The colour in the Display window for the section of track line visible in the editor.	
Event Marks	A unique marker placed at regular intervals in the survey.	

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Status

The Tools > Options - Status tab controls the status colours for soundings.

1. Select Status from the *Category* list.



2. Set the parameters described in the table below.

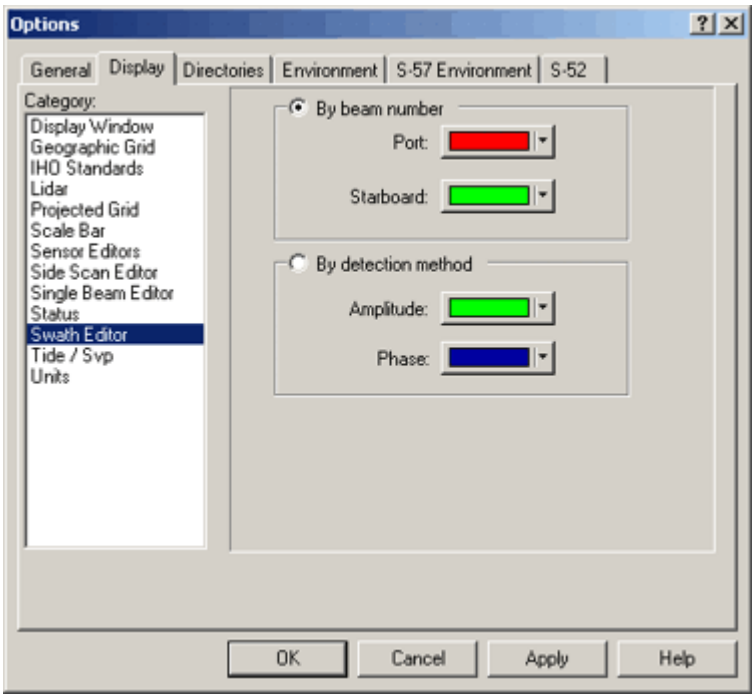
Field	Description	To assign a value...
Accepted	Colour for data that is flagged as ready for processing.	Select a colour from the colour palette, or create a custom colour from the standard Windows colour selector.
Rejected	Colour for data that is flagged as rejected and excluded from further processing.	
Reject - With Interpolation	Colour for attitude and navigation data that has been flagged as rejected but where associated soundings can still be processed.	
Reject - Break Interpolation	Colour for attitude and navigation data that has been flagged as rejected and where associated soundings are also rejected.	
Outstanding	Colour for soundings in the Subset Editor that have been flagged as needing further examination.	
Examined	Colour for soundings in the Subset Editor that have been flagged as examined and verified.	
Designated	Colour for the sounding that has been flagged in the Swath or Subset Editors as being the shoalest sounding on a feature.	

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Swath Editor

The Options - Swath Editor tab is for selecting colours for the Plan View display. Soundings can be coloured by location in the swath or by detection method. Soundings in the Rear, Side, and Profile Views are always coloured by their location in the swath.

1. Select Swath Editor from the *Category* list.



2. Set the parameters described in the table below.

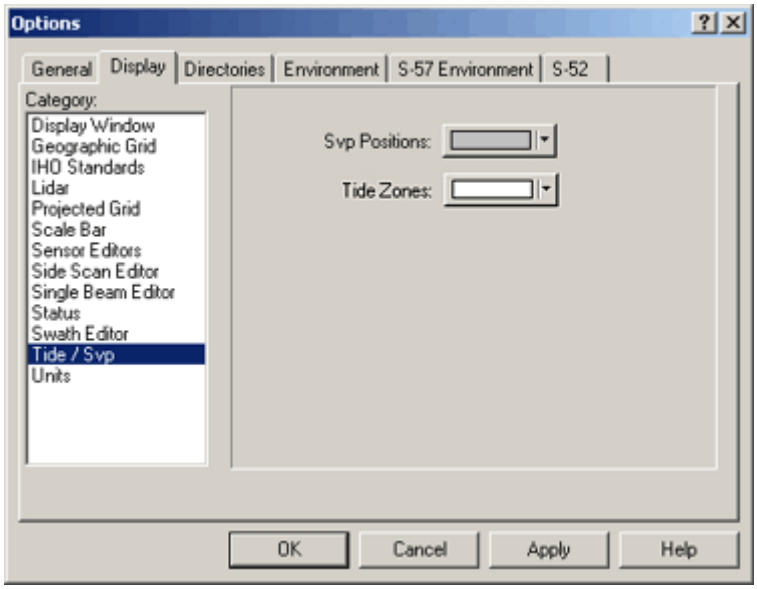
Field	Description	To assign a value...
By Beam Number	Colour soundings in the Plan View by location in the swath (port and starboard).	Optional: Click the *By Beam Numbe*r box so the port and starboard colour options are enabled. Use the *Port* and *Starboard* colour palettes to select a colour, or create a custom colour from the standard Windows colour selector.
By Detection Method	Colour soundings in the Plan by detection method used by the sonar.	Click the *By Detection Method* box so the amplitude and phase colour options are enabled. Use the *Phase* and *Amplitude* colour palettes to select a colour, or create a custom colour from the standard Windows colour selector.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Tide/SVP

The Options - Tide/SVP tab controls colour options for Sound Velocity Profile location symbols and tide zone lines.

1. Select Tide/SVP from the *Category* list.



2. Set the parameters described in the table below.

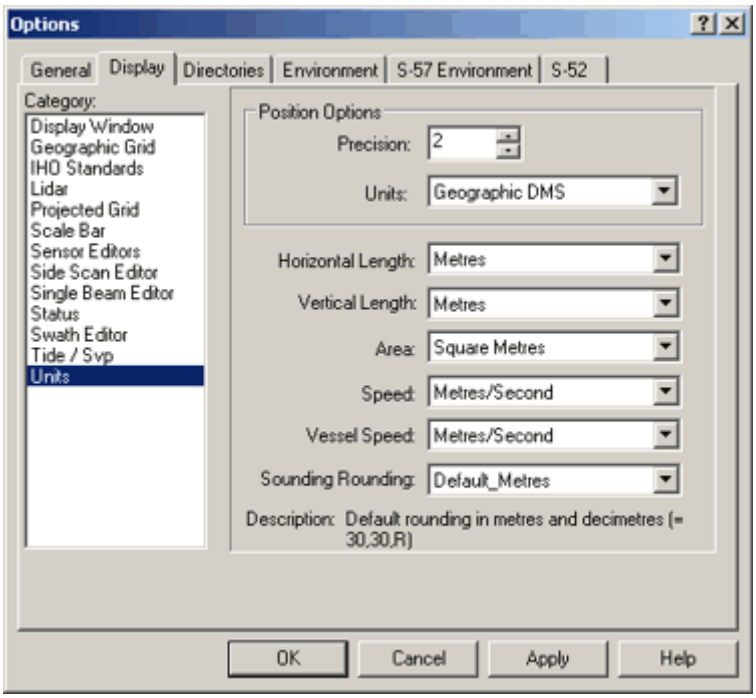
Field	Description	To assign a value...
SVP Positions	Colour for Symbols and text labels show the positions of sound velocity profiles.	Select a colour from the appropriate colour palette, or create a custom colour from the standard Windows colour picker.
Tide Zones	Colour for outline of tide zone areas.	

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Units

The Options - Units tab controls the measurement units that are used in all HIPS and SIPS applications (except the Field Sheet Editor).

1. Select Units from the *Category* list.



2. Set the parameters described in the table below.

Field	Description	To assign a value...
Units	There are five types of units: • Geographic DMS (Degrees, Minutes, Seconds) Example: 45-10-39.970N; 68-58-01.979W • Geographic DD (Decimal Degrees). Example: 45.170486; -68.964844 • Ground (ground units). Example: 548.73; 2116.24 • Pixel (units in pixels) • Geographic DM (Degrees, Minutes) Example: 45-10.970N; 68-58.97W	Select a unit from the pull-down menu.
Precision	Select how many places to the right of the decimal point that numbers are displayed for the cursor position.	Click the up or down arrow buttons beside the Precision field, or type a value in the field.
Horizontal Length	Length, height and area settings in metric or imperial units.	Select a unit setting from the pull-down menus.
Vertical Length		
Area		

Field	Description	To assign a value...
Vessel Speed	Speed units for Navigation Editor.	Select a unit setting from the pull-down menu.
Sounding Rounding	Select the depth and method that soundings are rounded/truncated.	Select a rounding/truncating rule from the drop-down list. A brief description of each rule is displayed when it is selected.

3. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Directories

The Directories tab lets you select the directories to be used by the system for finding and processing data.

The *Active Profile* options control where the directory settings are written to and read from in the Windows registry. The default setting for directory locations is *Administrator*. When this profile is selected, all environment settings are written to and read from the system-wide settings in the Windows registry. Changing these settings effects all users on the same machine who are using the *Administrator* option.

Only persons with a Power User or Administrator Groups account can make changes to directories while this option is selected. However, you can select your login name to override the *Administrator* settings and select the directories you want to access. When your user name is selected as the *Active Profile*, all environment settings are written to and read from your account settings in the Windows registry. Changes made using this option effects only the user that is currently logged in.

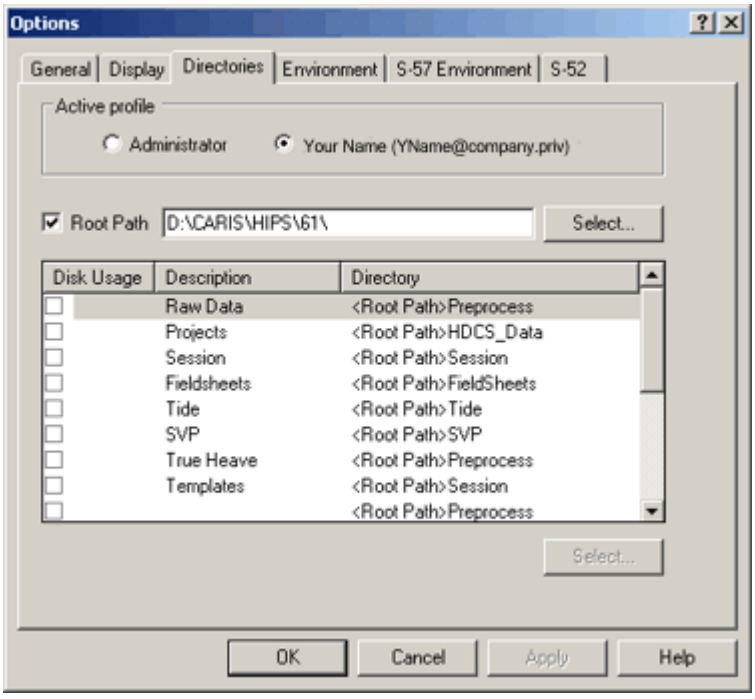
Options

Menu	Tools > Options

1. Select the Options command.

The Options - Directories dialog box is displayed.

2. Select the Directories tab.



3. Select the login name option to access directory locations in the application. The default setting is *Administrator*.

The *Root Path* option lets you select a file path to access a complete set of HIPS/SIPS directories. When this box is first checked, you are prompted to enter a path on your computer. This location is remembered by the program.

4. Select the *Root Path* check box if you want to switch to another set of directories on a network.
5. Type the path for the directory or enter the path by clicking **Select**.

The list in the directories tab is updated to show the new location. If a directory is not available then the directory at the previous location is used.

6. To change an individual directory, select it and double-click or click **Select**.

The Browse for Folder dialog box is displayed.

7. Click the expand icon to view the file that you want to select, if needed.
8. Select the file or folder.
9. Click **OK** to close the Browse for Folder dialog box.
10. The selected file path is displayed in the Options dialog box.
11. If you want to view the amount of disk use by a selected repository, click the appropriate check box.
12. The percentage of disk use is shown in the status bar.
13. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

Environment

The Environment tab controls the files or values used by HIPS and SIPS for ellipsoids, colour maps, feature codes, etc. All files are by default located in the System directory.

The *Active Profile* options control where the directory settings are written to and read from in the Windows registry. The default setting for directory locations is *Administrator*. When this profile is selected, all environment settings are written to and read from the system-wide settings in the Windows registry. Changing these settings effects all users on the same machine using the *Administrator* option.

Only persons with a Power User or Administrator Groups account can make changes to directories while this option is selected. However, you can select your login name to override the Administrator settings and select the directories you want to access. When your user name is selected as the *Active Profile*, all environment settings are written to and read from your account settings in the Windows registry. Changes made using this option will affect only the user that is currently logged in.

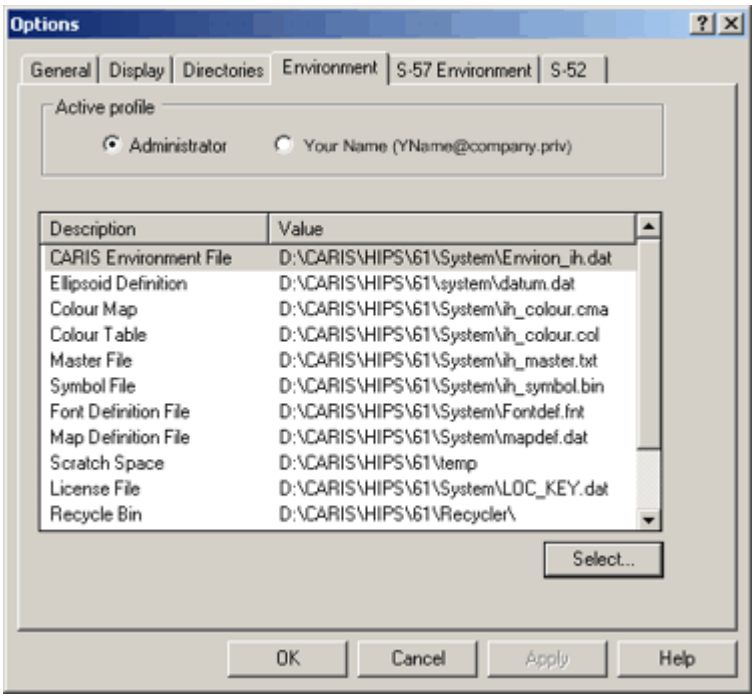
Options

Menu	Tools > Options

1. Select the Options command.

The Options - Environment dialog box is displayed.

2. Select the Environment tab.



3. Select your login name option to change files and values. The default setting is *Administrator*.

Change default environment settings

1. Select the file path displayed in the window.
2. Click **Select**.

The Open dialog box is displayed.

3. Select another file and click **Open**.

The selected file is displayed in the Environment tab.

4. Click **Apply** to implement the settings without closing the dialog box, or click **OK** to save the settings and close the dialog box.

S-57 Environment

The S-57 Environment tab controls the files and values used for displaying S-57 data in the Display window. The default directory for S-57 display files is \HIPS\System\S57Config.

The *Active Profile* options control where the directory settings are written to and read from in the Windows registry. The default setting for directory locations is *Administrator*. When this profile is selected, all environment settings are written to and read from the system-wide settings in the Windows registry. Changing these settings effects all users on the same machine using the *Administrator* option.

Only persons with a Power User or Administrator Groups account can make changes to directories while this option is selected. However, you can select your login name to override the *Administrator* settings and select the directories you want to access. When your user name is selected as the *Active Profile*, all environment settings are written to and read from your account settings in the Windows registry. Changes made using this option will effect only the user that is currently logged in.

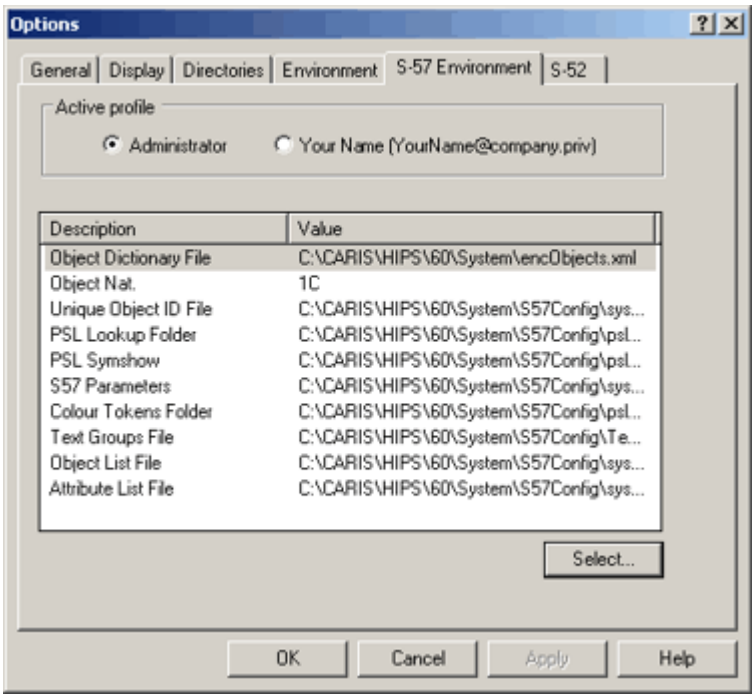
Options

Menu	Tools > Options

1. Select the Options command.

The Options - S-57 Environment dialog box is displayed.

2. Select the S-57 Environment tab.



3. Select your login name option to change files and values. The default setting is *Administrator*.

1. Select the file path displayed in the window.
2. Click **Select**.

The Open dialog box is displayed.

3. Select another file and click **Open**.

The selected file is displayed in the S-57 Environment tab.

4. To change back to the original file path that was used when HIPS and SIPS was installed, click **Default**.
5. Click **Apply** to implement the settings without closing the dialog box, or click **OK** to save the settings and close the dialog box.

S-52 Display

The S-52 Display tab controls the appearance of S-57 Ed. 3.1 data shown as background data in the Display window

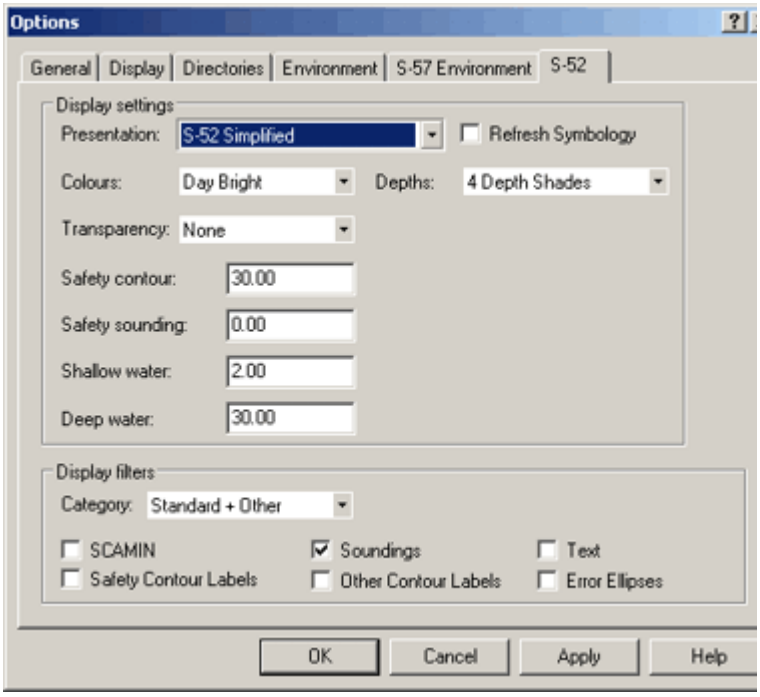
Options

Menu	Tools > Options

1. Select the Options command.

The Options - S-52 dialog box is displayed.

2. Select the S-52 tab.



3. Set the parameters described in the table below..

Field	Description	To assign a value...
Presentation	Display S-57 and HOB data in S-52 or INT1 format.	Select a presentation format from the drop-down list.
Refresh Symbology	Automatically refresh the display to show the symbology associated with the selected presentation library.	Select the check box to make this option active.

Field	Description	To assign a value...
Colours	Select the time-of-day colour scheme in which to display S-57 data: Day Bright, Day Dark, Day White, Dusk, Night Filtered, Night No Filter.	Select the appropriate item from the drop-down list.
Points	Select *Simplified* (default), which means to symbolize using the S-52 Presentation Library; or *Full Chart*, which means to symbolize using IHO (International Hydrographic Organization) chart specifications (the way an IHO-compliant paper chart appears).	
Depths	Select the number of depth shades to use: 2 or 4 (default). This value works in conjunction with Safety Contour Depth, Shallow Water and Deep Water	
Areas	Select *Plain Boundaries* (default), which means to symbolize using the S-52 Presentation Library; or *Symbolized Boundaries*, which means to symbolize using IHO chart specifications (the way an IHO-compliant paper chart appears).	
Transparency	Select a transparency setting (none, 25, 50, 75, or 100 percent) for polygon fills in the display.	
Safety Contour	The areas that have sufficient depth for safe navigation. The safety contour is symbolized as a thick solid line. It also defines the boundary of shallow and deep-water areas for the purpose of area symbolization.	Type a value in the appropriate field.
Shallow Water	The contour boundary between the two darker colours used for water shallower than the Safety Contour.	
Safety Soundings	The safety depth is used in the symbolization of soundings. Soundings whose depth is less than the safety depth are displayed in bold text.	
Deep Water	Define the contour boundary between the two lighter colours used for water deeper than the Safety Contour.	
Categories	Pre-defined categories of objects to be displayed	Select a category from the drop-down menu.

Field	Description	To assign a value...
SCAMIN	Select this option to filter features from the Display window display that have SCAMIN values less than the display scale. These features are however, still drawn in the selection/superselection colours if they are selected or superselected.	Select or clear the check box to turn the display on or off
Safety Contour Labels/ Contour Labels	Show labels on safety contours and/or other contours.	
Soundings	Control the display of soundings.	
Text	Control the display of text.	
Error Ellipses	Enable the display of the Errell (error ellipse) attribute.	

4. Click **Apply** to implement the changes without closing the dialog box, or **OK** to implement the changes and close the dialog box.

4

Data Directories

Introduction

The structure of the HIPS and SIPS data directories is designed to maximize data access efficiency during processing. By definition, these directories include only those used to store raw or processed data and products, not the HIPS and SIPS applications nor their associated system files and libraries. The data directories are listed in the following table.

Data Type	Default Directory Location
Projects	..\Caris\Hips\HDCS_Data
Raw data	..\Caris\Hips\Preprocess
Session files	..\Caris\Hips\Session
Field sheets	..\Caris\Hips\Fieldsheets
Tide	..\Caris\Hips\Tide
SVP	..\Caris\Hips\Svp

For reasons of consistency with older code contained within HIPS and SIPS, directory names and paths should not contain any spaces or special Windows characters (\, /, :, *,?,",<,>, and |).

Projects

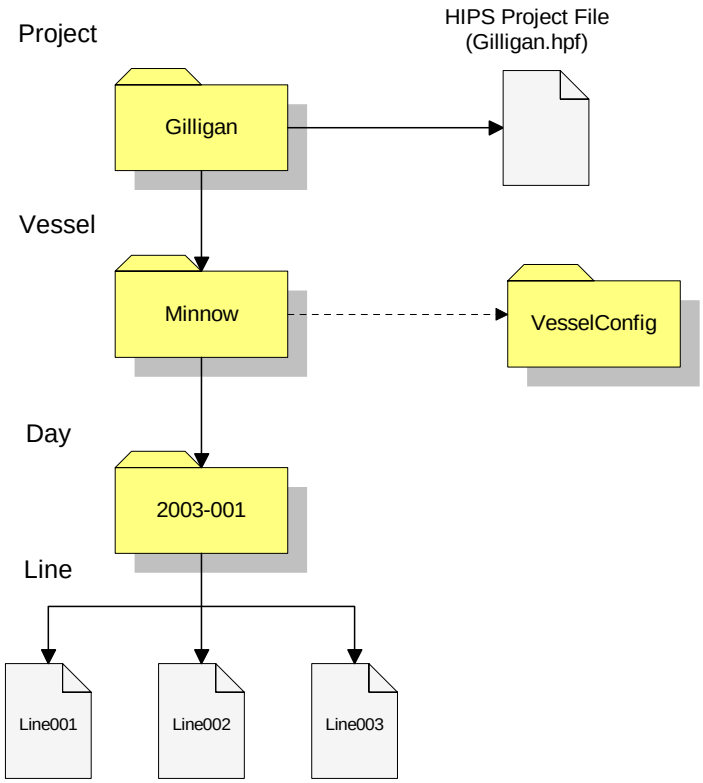
The Projects directory holds all of the converted data. The Project directory contains two main components:

- HIPS and SIPS data ("HIPS AND SIPS DATA" ON PAGE 109)
- Vessel files ("VESSELCONFIG DIRECTORY" ON PAGE 110)

HIPS and SIPS Data

Converted data in HIPS and SIPS is stored in a Project/Vessel/Day/Line (P/V/D/L) structure. Each survey project is divided into one or several vessels (for example, survey launches), contributing to the same project. The data can be from different generations of surveys, or different hardware collection platforms. Each vessel folder is divided into separate survey days and each survey day contains one or more survey lines.

Below is an outline of the P/V/D/L structure for a project called Gilligan with a vessel called Minnow. The project contains three line files.



In the Project directory, in addition to the folder(s) containing the vessel(s), there is also a HIPS Project file (HPF) that contains the projection information used by the HIPS and SIPS interface to define the default projection for that project. When HIPS and SIPS accesses the navigation data (which is stored in unprojected latitude and longitude), the data is projected using the projection stored in this file. The HPF is given the same name as the project. For more information on creating a new project see "DEFINE PROJECT" ON PAGE 150.

The name of the vessel in the P/V/D/L path provides the link to the actual HIPS Vessel File and therefore must be spelled the same as the HVF in the VesselConfig directory. The data cannot be processed if the HVF is not present or if the vessel name in the P/V/D/L path is misspelled.

The survey day in the P/V/D/L structure is composed of two components separated by a hyphen—the four-digit year and the three-digit Julian date. The date used in the Day directory name does not actually play a role in data processing—it is used as a convenient mechanism for organizing survey data.

When a new project is created, this directory structure is built and the HPF is generated. The vessel file must exist in the VesselConfig directory at the time the project is defined, however it can be modified at any time (providing the name is not changed).

VesselConfig Directory

The VesselConfig directory contains the HIPS Vessel File (HVF) used in the vessel component of the P/V/D/L path. The name of this directory must not be changed and it always has to be located inside the Raw Data directory on the same level as the project directories.

For more information about the HVF see "VESSEL FILES" ON PAGE 5-2.

Raw Data

The raw data directory is used to store unprocessed files that are used as input into HIPS via the Conversion Wizard. Unprocessed files can be stored in any order and in any location on your system. However it is recommended that you store these files in a folder structure that mimics the P/V/D/L structure of the Projects folder.

Not only does the P/V/D/L structure provide a consistent path for data retrieval. You can also take advantage of a feature in the Conversion Wizard that lets you open for conversion at any level of the P/V/D/L folder structure and maintain that structure through conversion.

For example, a new project called "Gilligan" is created by the New Project wizard without adding a vessel or survey days to the project definition.

Raw data is then converted to HIPS and SIPS format through the Conversion Wizard. The File Selection Type is set to "Project" in Step 2 of wizard. The converter then knows that data is stored in the P/V/D/L structure and that you are loading from the Project level. All folders on the next level are interpreted as vessel names and the level after that as survey days.

Each vessel name must exist in the VesselConfig directory, otherwise the converter will report errors. In this example, there is only one vessel folder, called "Minnow" and it contains several survey days. There happens to be a HVF in the VesselConfig directory called "Minnow," so the converter creates the vessel folder beneath the project name "Gilligan" in the Project directory and also all of the survey days that are necessary. If the survey days or the vessel are already present, the converter uses them.

Finally, all of the lines found are converted.

To convert raw data that is not stored in the P/V/D/L structure, simply set the file selection type to "Raw" and select the files individually.

For more information on converting data to HIPS/SIPS format see “Rename Day and Line Folders” on page 156.

SVP and Tide

SVP

The SVP directory contains all of the sound velocity profile files. They can be stored in subdirectories of this directory if desired.

Tide

The Tide directory contains all of the tide files. They can be stored in subdirectories if desired.

5

HIPS Vessel Editor

Vessel Files

The HIPS Vessel Editor is an application for viewing and editing HIPS vessel information. Vessel information is contained in the HIPS Vessel File (HVF).

The HVF is an Extensible Mark-up Language (XML) file that describes the installation and calibration of equipment installed on the survey vessel. The information in the HVF is used by multiple processes including merging, mosaicking, and calculating sounding uncertainty.

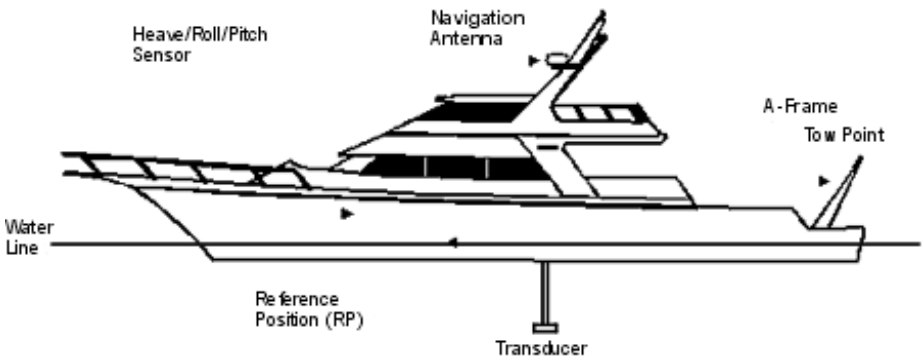
The HVF is divided into a number of distinct sections, each describing one type of sensor. The sections are time-tagged and multiple entries can be defined for different time periods. The HVF for a ship is stored in the default directory …\Hips\HDCS_Data\VesselConfig.

NOTE: In previous versions of HIPS and SIPS, a text file called the Vessel Configuration File (VCF) was used in place of the HVF. VCF files can still be opened in the Vessel Editor and edited, but when the file is re-saved it is converted to an HVF format.

The following sections describe common features in the HVF.

Vessel Coordinate System

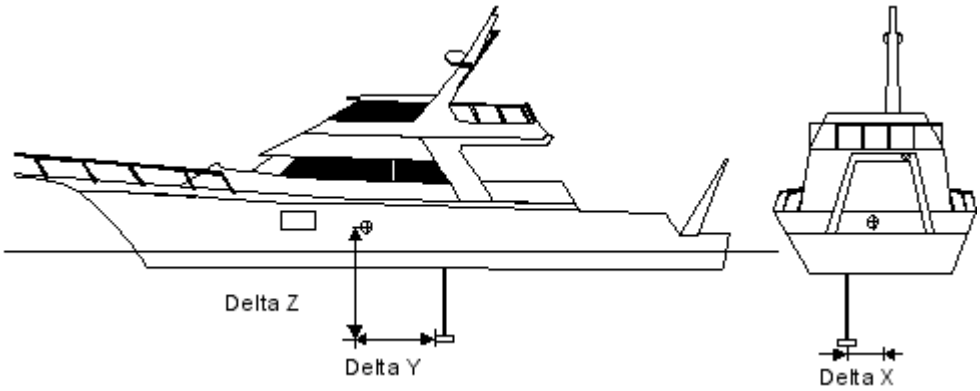
Vessel configuration is based on an instantaneous three-dimensional coordinate system, which is used to locate sensors and other equipment. Below is an example of various sensor equipment locations on board a vessel.



The vessel coordinate system locates equipment with an X-Y-Z axis that uses a Reference Position (RP) as a point of origin. The location of the RP depends upon the type of sonar system used in the survey.

The axes are defined as follows:

- The Y-axis is oriented along the vessel's fore/aft axis, positive forward.
- The X-axis is oriented along the vessel's port/starboard axis, perpendicular to the Y-axis, positive to the starboard.
- The Z-axis is perpendicular to the X-Y plane, and positive into the water.



To ensure that the correct signs are used when entering X-Y-Z sensor offsets into the Vessel Editor, the sensor location must be described relative to the Reference Point. Imagine standing at the location of the Reference Point and describing how far and in what direction the sensor is away from you.

The location of the vessel's coordinate system within the local fixed coordinate system is determined by the navigation system and tide. The orientation of the coordinate system is defined by the vessel's gyro and attitude sensors (pitch and roll). Gyro, pitch and roll observations are defined as follows:

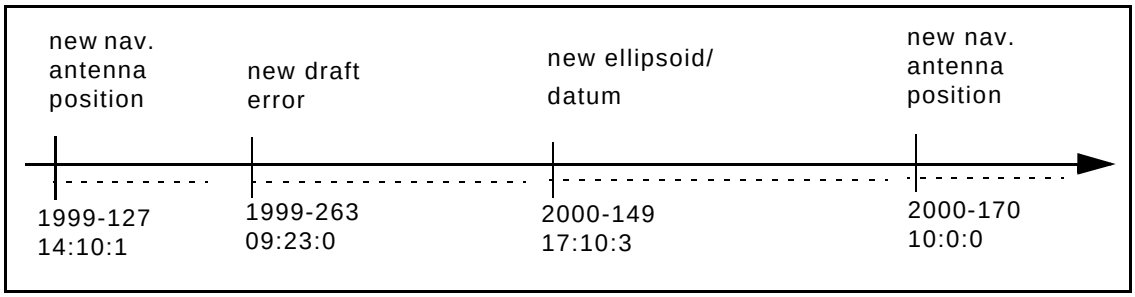
- A positive gyro observation is defined as the clockwise rotation of the vessel (from 0 and 360 degrees) within the navigation coordinate system.
- A positive pitch is observed when the bow of the vessel is down (bow down).
- A positive roll is observed when the starboard side of the vessel is up (starboard up).

Reference Position

The Reference Position (RP) is a location from which all other offsets are derived. Its location is usually at the centre of the ship's gravity. This position is used as the centre of the ship's rotation when applying the pitch/heave/roll parameters, and must be applied as accurately as possible.

Time Stamp

A vessel's configuration changes over time: Different draft settings may be used in a survey, or the position of the transducers on a boom may change, or a different ellipsoid is used. A time stamp is used to keep track of these changes. The time stamp records the time from which a configuration is valid and is recorded with each entry to the HVF.Below is an example of time stamps and vessel configuration.



Time in HIPS is based on the recorded reference time. No distinction is made for UTC.

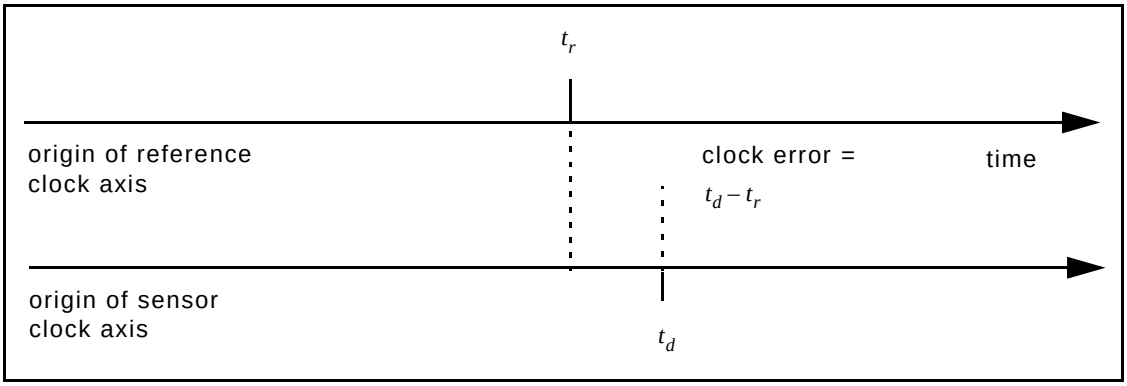
Time Correction

In the HVF, most sensors have a Time Correction field that contains the time difference between the sensor clock and the reference clock. Sensors are time tagged and all clocks are compared to the reference clock so sensor readings can be synchronized.

The Time Correction field is specified in seconds and is positive if ahead of the reference clock:

`Time Correction = Recorded Time - Reference Time`

The following figure shows time correction with respect to the sensor clock.



Working With the Vessel Editor

Open the Vessel Editor

The Vessel Editor is launched from the HIPS and SIPS interface.

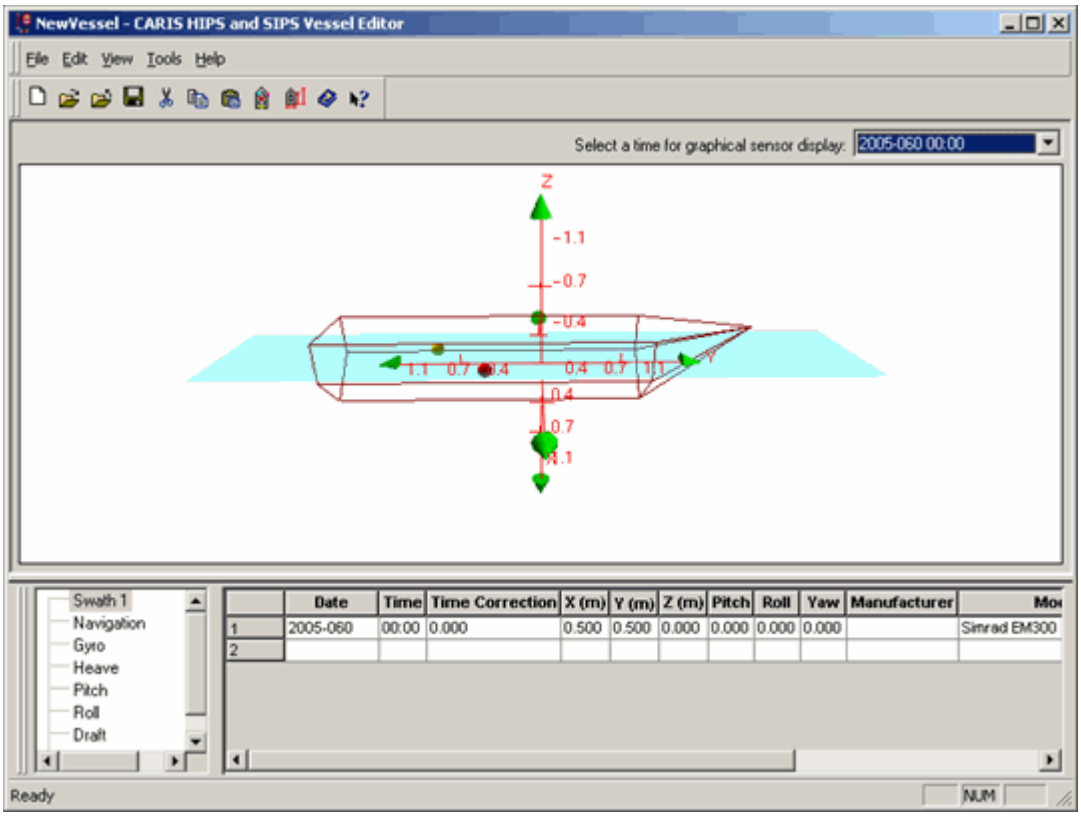
Vessel Editor

Menu	Edit > Vessel Editor
Tool	

1. Select the Vessel Editor command.

The Vessel Editor is displayed.

Below is an example of the Vessel Editor with a vessel file displayed.



Overview

The vessel is displayed in 3-D format. The display can be manipulated using the 3-D display compass. Sensor location is represented by a coloured dot or line.

Above the vessel outline is a drop list containing all the unique time stamps in the vessel file. Because the graphical display can only display the configuration at a single point in time, select one of the time stamps for viewing the vessel outline.

Below the outline are the tables where you enter information on the sensors used in the survey.

Open, Close and Save Files

Open a saved file in the Vessel Editor.

1. Select a Open command.

Menu	File > Open
Tool	
Key	<Ctrl + O>

The Open File dialog box is displayed.

2. Click a file so it is displayed in the *File Name* text box.
3. Click **OK**.

The file opens in the Vessel Editor with the vessel's outline displayed in the editor, and a list of active sensors displayed in the Sensors list box.

> Vessel Configuration Files, created in previous versions of HIPS and SIPS, can be opened and edited in the Vessel Editor. The files are automatically re-saved as HIPS Vessel Files.

Save changes to an already existing file.

1. Select a Save command.

Menu	File > Save
Tool	
Key	<Ctrl + S>

Changes made to file are automatically saved.

To save the file under a new name, select **File > Save As**.

1. Select the Close command.

Menu	File > Close
Tool	
Key	<Ctrl + F4>

You are prompted to save changes made in the Vessel Editor.

2. Click **OK**.

Create a text file listing the parameters entered in Vessel Editor.

1. Make sure a HVF is open in the editor.

2. Select the Vessel Report command.

Menu	Tools > Vessel Report

The Save As dialog box is displayed. The default directory is Hips\6.1\HDCS_Data\VesselConfig.

3. Type a name for the file in the *File Name* field.

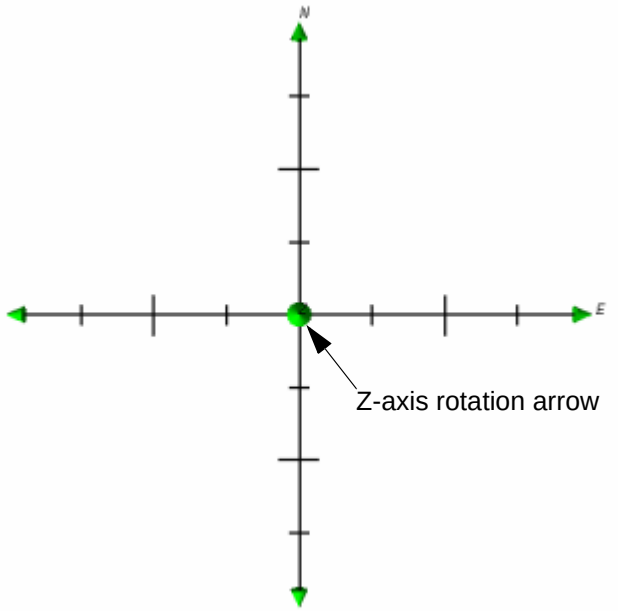
4. Click **Save**.

3-D Controls

The display in the editor can be rotated, zoomed and panned using the compass, keyboard, and mouse.

Compass

Rotation along the X-Y-Z axis is handled by moving the arrow heads at the ends of the compass.



1. To rotate the image along the Z-axis, position the cursor on the middle arrow head, press and hold the mouse button, and drag the cursor upward so that the image is rotated. The compass can be rotated 90º.
2. To rotate the image along the X-Y axis, position the cursor on any of the other arrow heads, press the mouse button and drag the arrow head in a left-right or up-down direction. The compass can be rotated 360º.
3. Alternatively, the image can be rotated in the horizontal plane by grabbing the Z-arrow, then pressing the <Ctrl> key and the left-mouse button and moving the cursor up and down in the view. Rotation around the Z-axis can be performed by pressing the <Shift> key and moving the cursor left and right in these windows.

Pan

There are three pan options:

- To pan along the X-Y axis: place the cursor in the 3-D area, press the middle mouse button and drag the display to a new location.
- To pan along the Z-axis: place the cursor in the 3-D area, press and hold both the <Shift> key and middle mouse button and move the cursor in the window.
- To centre the 3-D Display window on a selected point: position the cursor over the point and click the middle mouse button.

Zoom

The zoom option is similar to pan

1. Press and hold both the <Ctrl> key and right-mouse button while dragging the mouse up or down to zoom in or zoom out.

Create Vessel Outline

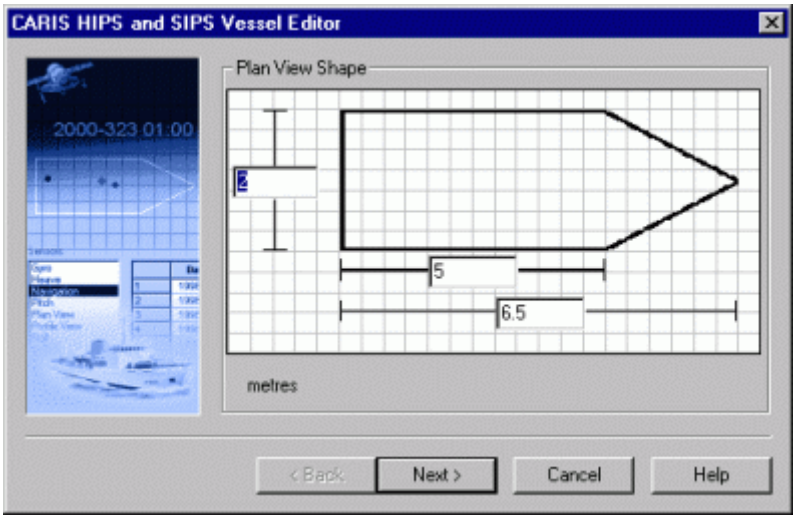
The Vessel Shape command creates a 3-D vessel outline that is displayed in the Editor. You may want to make revisions to an existing vessel outline or create a new outline if one does not exist in the HVF.

The Vessel Shape command opens three dialog boxes that define the length, width and height of the vessel.

1. Select the Vessel Shape command.

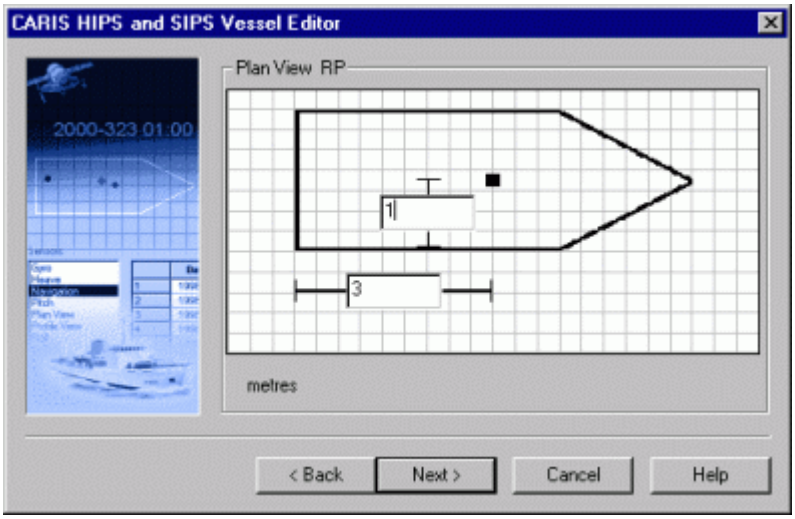
Menu	Edit > Vessel Shape
Tool	

The Vessel Shape - Step 1 dialog box is used for entering width and length.



1. Enter a width for the vessel (port to starboard).
2. Enter a length for the vessel (stern to the base of the bow).
3. Enter a length for the vessel (stern to the tip of the bow).
4. Click **Next**.

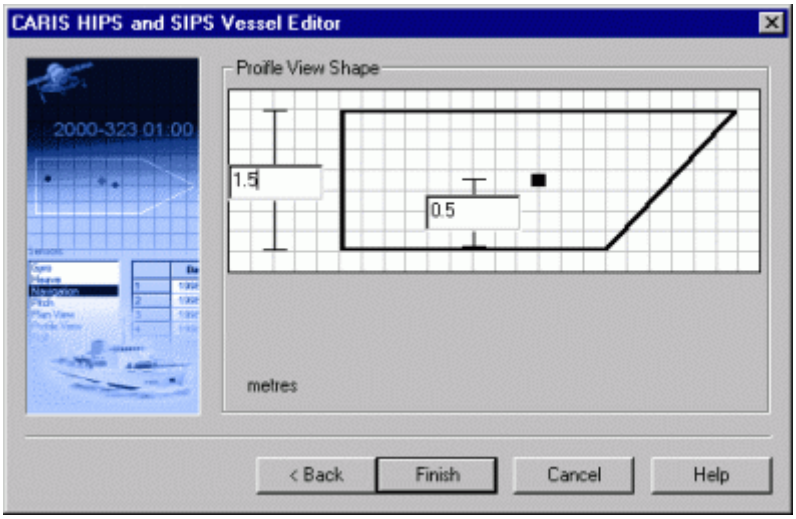
The Vessel Shape - Step 2 dialog box is displayed.



The length and width to the Reference Position (RP) is calculated from the values you entered in the previous dialog box. If you have to change these values, complete the following steps.

1. Type the length (from the stern to RP).
2. Type the width (from the starboard side to the RP).
3. Click **Next**.

The Vessel Shape - Step 3 dialog box sets the height measurements for the vessel.



1. Type the height value in the box.
2. Type the height of vessel RP from the keel.
3. Click **Finish**.

The Vessel Editor opens with the vessel outline displayed in the interface.

Options Dialog Box

Default ellipsoid

Select a ellipsoid for a new vessel file. The ellipsoid should be the same as the one used in the survey. The ellipsoid is displayed in the Navigation section of the editor.

The list of available ellipsoids is maintained in the datum.dat file referenced by the environment variable, uslXdatum. By default, this file is located in the System directory.

1. Select the Options command.

Menu	Tools > Options

The Options dialog is displayed.

2. Select the Options - General tab.
3. Select an ellipsoid from the pull-down menu.
4. Click **Apply** to apply and save the setting to the file.
5. Click **OK** to close the dialog box.

Colours

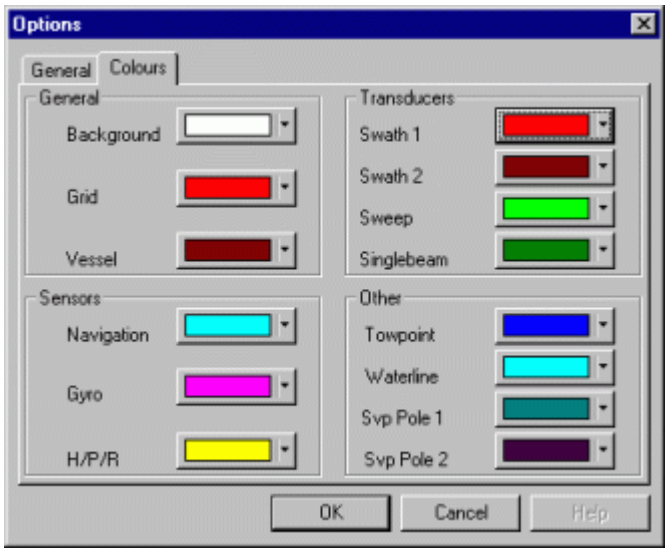
The locations of sensors and other equipment are shown by a colour markers. You can select another colour for a sensor marker.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.

2. Select the Options - Colours tab.



3. Select a colour from the pull-down palette, or create a custom colour from the standard Windows colour picker.
4. Click **Apply** to activate the changes but not close the Options dialog.
5. Click **OK** to activate the changes and close the Options dialog.

Exit the Vessel Editor

Close the Vessel Editor and all files.

1. Select an Exit command.

Menu	File > Exit
Key	<Alt + F4>

You are prompted to save any changes that have not yet been saved.

2. Click **OK**.

The Vessel Editor closes.

Create a New HVF

Menu	File > New
Tool	

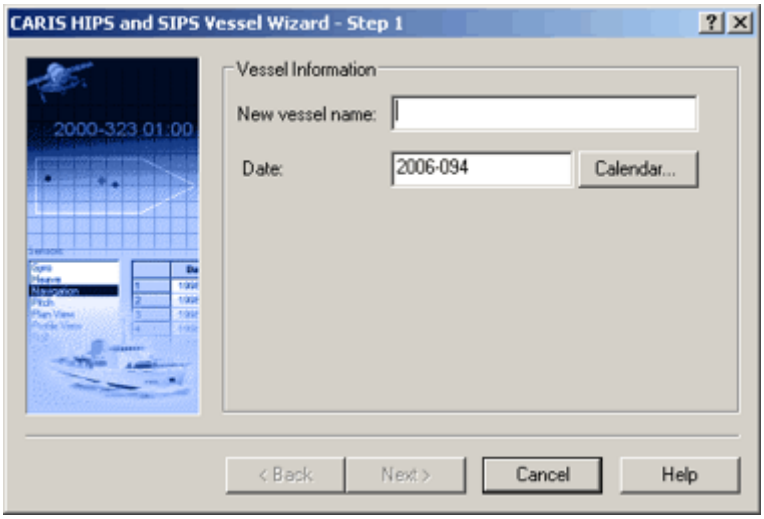
Use the HIPS and SIPS Vessel Wizard to set the parameters needed to create a new file. Additional information is entered in the Vessel Editor after the file has been created.

1. Select the New Vessel File command.

The Vessel Wizard Step 1 dialog box is displayed.

(The dialog boxes displayed by the wizard are contingent on the type of sonar used in the survey.)

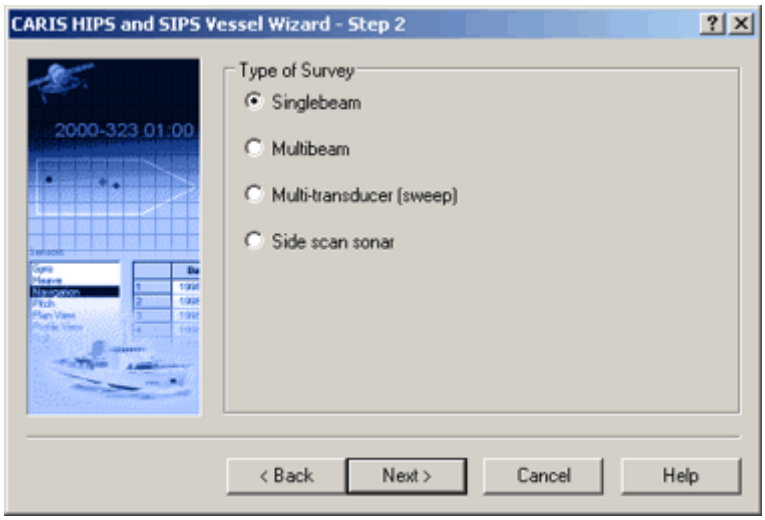
Vessel Information



1. Type a name similar to the vessel used in the survey. Make sure the name does not include any spaces.
2. Click the **Calendar** button to insert the year and date from which the vessel file is valid.
3. Click **Next**.

Type of Survey

The second dialog box prompts you to enter the sonar type used in the survey.



There are four sonar types:

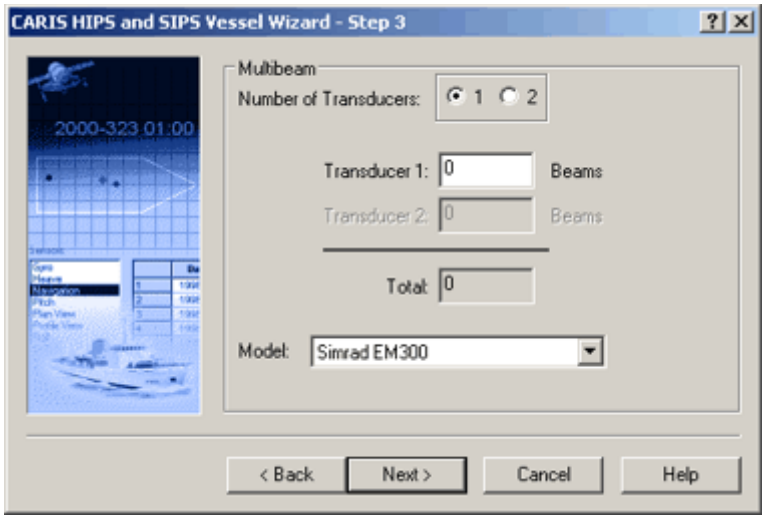
- *Singlebeam*
- *Multibeam*
- *Multi-transducer*
- *Side Scan Sonar*

1. Select a sonar type by clicking the appropriate check box.
2. Click **Next** to go to the next step in the Vessel Wizard, or if you selected *Side Scan Sonar*, click **Finish**.

Single Beam

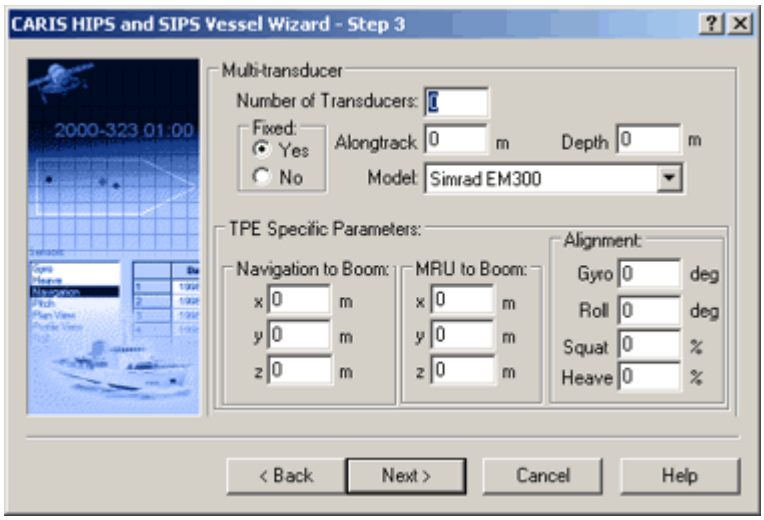
If you select Single Beam as the type of survey in Step 2, then the remaining steps in the creation of a new HVF are "Motion Sensors" on page 129 and "Configuration Options" on page 130.

Multibeam



1. Select the appropriate check box to indicate if one or two transducers are used in the survey.
2. Enter the number of beams in each transducer.
3. Select the sonar model.
4. Click **Next** to open the Motion Sensor step dialog box.

Multi-Transducer



1. Enter the *Number of Transducers* on a boom.
2. Select *Yes* if the transducers are mounted to a fixed boom or to the hull of the vessel, or select *No* if the transducers are on a movable (decoupled) boom.
3. Type the default *Alongtrack* value of all the transducers relative to the ship's reference point.
4. Type the default *Depth* of all the transducers relative to the ship's Reference Point.
5. Select the sonar model from the *Model* drop-down list.

The following parameters are necessary to calculate Total Propagated Error.

6. Type the distances from the positioning system to the centre of the boom in the *Navigation to Boom* X-Y-Z fields.
7. Type the distance from the motion recording unit to the centre of the boom in the *MRU to Boom* X-Y-Z fields.

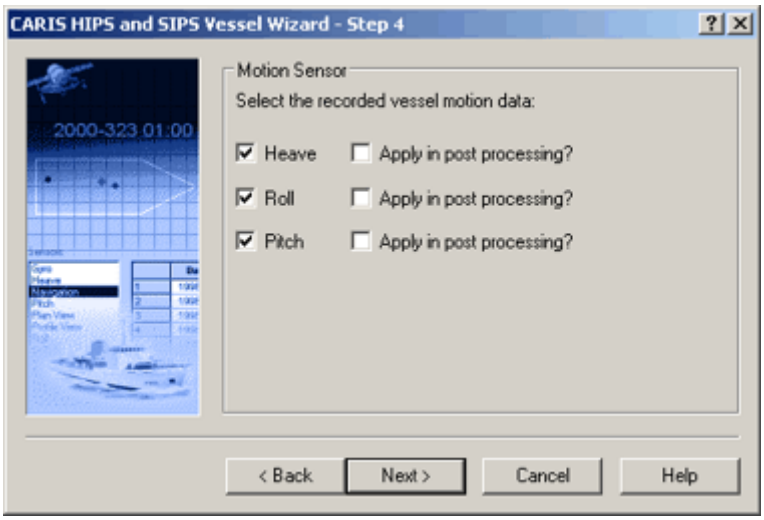
The *Roll* and *Gyro* misalignment values are needed for fixed-boom system. However, for a decoupled boom, only a *Gyro* misalignment value is needed.

The *Squat* and *Heave* percentage values are transducer responses to overall heave and squat. For fixed-boom transducers this should be 100% and less than 100% for decoupled-boom transducers.

8. Type a degree offset in the *Roll* and *Gyro* fields.
9. Type a percentage value in the *Squat* and *Heave* fields.
10. Click **Next** to open the Configuration Options step.

Motion Sensors

This dialog box determines what attitude sensors are displayed in the HVF.



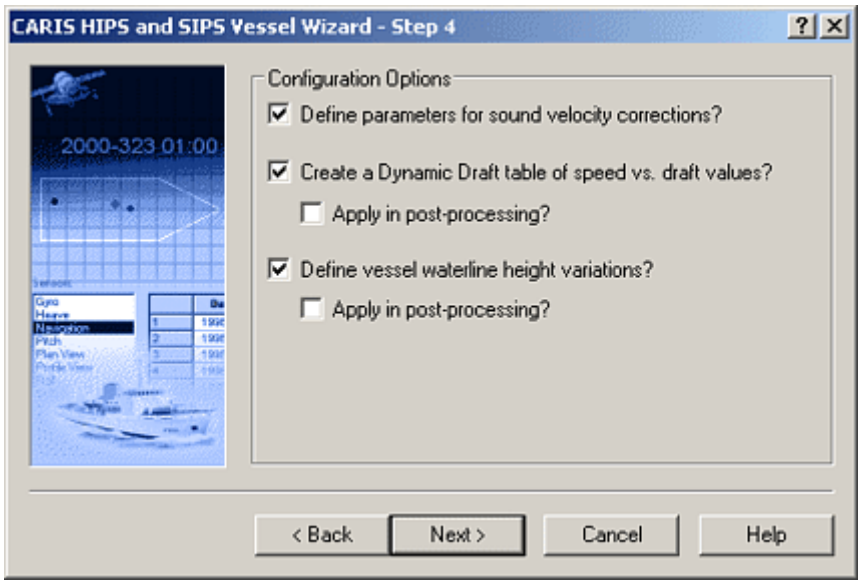
1. Create entries for any (or all) of the following sensors by clicking the appropriate box:
   - *heave*
   - *pitch*
   - *roll*

The *Apply in Post Processing* option means that the data is applied in the Merge process or in Sound Velocity Correction.

2. Click any of the check boxes if you want to apply the attitude sensor data in post processing.
3. Click **Next**.

Configuration Options

The Configuration Options dialog box is displayed


CARIS HIPS and SIPS Vessel Wizard - Step 4
2000-323 01:00
Configuration Options
Define parameters for sound velocity corrections?
Create a Dynamic Draft table of speed vs. draft values?
Apply in post-processing?
Define vessel waterline height variations?
Apply in post-processing?
< Back
Next >
Cancel
Help

This dialog box lets you enable three options:

- the SVP (Sound Velocity Profile) pole parameters
- dynamic draft values
- waterline height variation

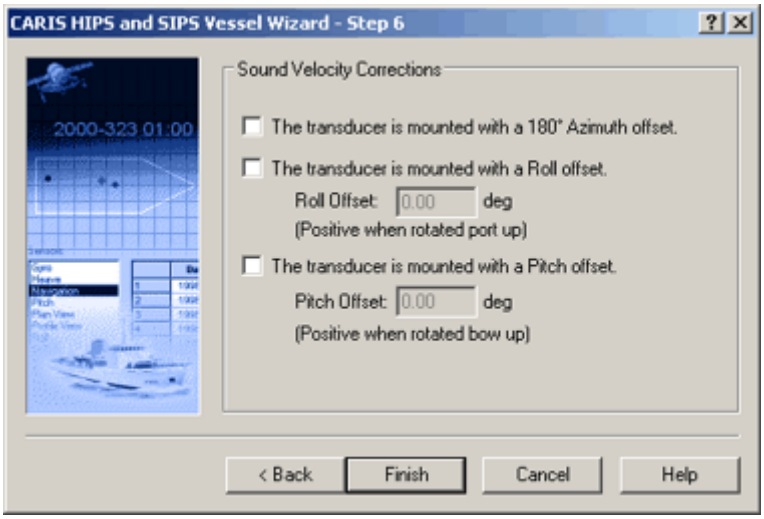
1. To include sound velocity corrections, click the *Define Parameters for Sound Velocity Corrections* check box.
2. To apply dynamic draft values to vessel speeds, click the *Apply Dynamic Draft* check box.

A draft-versus-speed table is created in the Vessel Editor. You can enter values in this table once the vessel file is created.

3. If you want to apply long period variations in the waterline height in processing data, click the *Define Vessel Waterline Height Variation* check box.
4. Click the appropriate *Apply in Post Processing* check boxes, as needed.
5. If you selected the *Define Parameters for Sound Velocity Corrections* check box, click **Next**, or click **Finish** if did not select this option.

Sound Velocity Corrections

The Sound Velocity Corrections dialog box is displayed only if you selected the *Define parameters for sound velocity corrections* check box in the previous dialog box.


CARIS HIPS and SIPS Vessel Wizard - Step 6
2000-323 01:00
Sound Velocity Corrections
The transducer is mounted with a 180° Azimuth offset.
The transducer is mounted with a Roll offset.
Roll Offset: 0.00 deg
(Positive when rotated port up)
The transducer is mounted with a Pitch offset.
Pitch Offset: 0.00 deg
(Positive when rotated bow up)
< Back
Finish
Cancel
Help

This dialog box records any transducer pole offsets.

Transducer head offsets must be taken into account when the SVP is applied to the data.

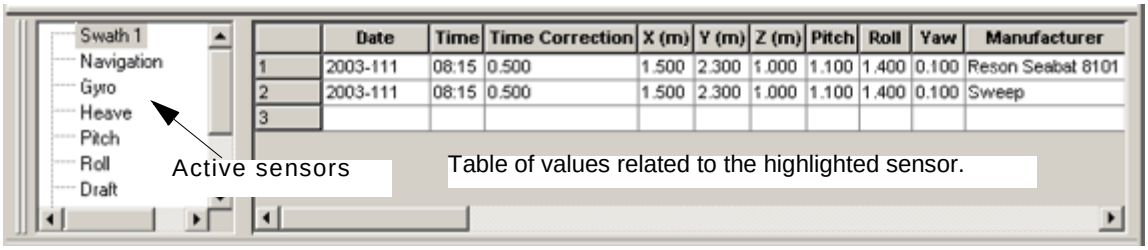
1. Select the 180-degree Azimuth box if the transducer is reverse mounted.
2. If the transducer is mounted with a large roll offset for use under wharves or along banks, then select the *Roll Offset* check box
3. Type the amount of offset (in degrees) in the *Roll Offset* box.
4. If the transducer is mounted with a large pitch offset such that it is pointed forward or backwards, then select the *Pitch Offset* check box.
5. Type the amount of offset (in degrees) in the *Pitch Offset* box.
6. Repeat the above steps for the second transducer if there are two poles.
7. Click **Finish**.

A new HIPS Vessel File is created. You can create an outline for the vessel (see "CREATE VESSEL OUTLINE" ON PAGE 121).

Edit Sensor Configuration

The active sensors shown in the Vessel Editor list (shown below) are determined by the sensor and other parameters you selected when creating the vessel file.

If you want to add data for a sensor not already included in the vessel file, you can add a sensor to the list and enter its data. You can also remove a sensor and its values from the vessel file.



	Date	Time	Time Correction	X (m)	Y (m)	Z (m)	Pitch	Roll	Yaw	Manufacturer
1	2003-111	08:15	0.500	1.500	2.300	1.000	1.100	1.400	0.100	Reson Seabat 8101
2	2003-111	08:15	0.500	1.500	2.300	1.000	1.100	1.400	0.100	Sweep
3										

In general, if sensor offsets or calibration values have already been applied to logged data during data acquisition, then the same offsets and calibration parameters must not be entered in the Vessel Editor.

Also, if compensation for heave, pitch, and roll has already been made to the recorded sounding data during data acquisition, for example as in Simrad data, then it must not be applied again.

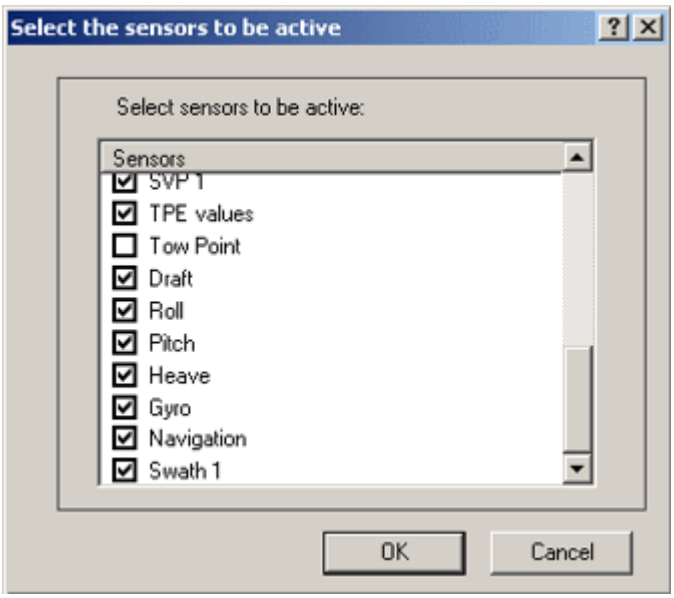
Toggle data fields

To add or remove sensors from the list of active sensors

1. Select the Active Sensors command.

Menu	Edit > Active Sensors
Tool	

The Active Sensors dialog box is displayed.



If a box is checked, the sensor is included in the vessel file and displayed in the Editor table.

2. Make sure a sensor box is checked to enable the sensor, or cleared to remove the sensor.
3. Click **OK**.

The list is refreshed to show the selected active sensors.

The following sections explain the individual sensor data fields.

Swath

The swath sonar, also known as multibeam, typically has beams that are formed in a fan shape, radiating from the centre of the transducer.

1. Click *Swath 1* in the Sensors list box so the selection is highlighted and the swath data fields are displayed.
2. Type data (as needed) in the following fields:
   - Date: The year and Julian day of current swath time stamp.
   - Time: The hour and minute of the current swath time stamp.
   - Time Correction: The time correction value.

The X-Y-Z fields set the location of the transducer from the Reference Point (0).

- *X*: The athwart-ship distance of the transducer, positive to starboard.
- *Y*: The along-ship distance of the transducer, positive to the bow.
- *Z*: The vertical distance of the transducer, positive into the water.

The Pitch, Roll and Yaw fields refer to misalignment of the transducer, during mounting, from the vessel coordinate system.

- *Pitch*: The offset is positive when rotating the transducer towards the bow (bow up).
- *Roll*: The offset is positive when rotating the transducer away from starboard (starboard down).
- *Yaw*: Indicates the horizontal rotational offset and is positive for a clockwise rotation.
- *Manufacturer*: The maker of the transducer.
- *Model*: The make of the transducer. The model information is necessary for the calculation of Total Propagated Error.
- *Serial Number*: The serial number of the transducer.

Since the Simrad data read into HIPS is already corrected, the Swath values in the HVF must be set as follows:

- The X/Y/Z offsets are zero because the Simrad data acquisition has already applied static draft and shifted the swath profile to the vessel reference point.
- The Roll / Pitch / Yaw transducer mounting rotations are typically zero because the Simrad data acquisition has already applied patch test calibration results.

Navigation

The navigation section of the HVF describes the location of the navigation source (for example, antenna or motion sensor). This section is used as a link between the fixed positioning coordinate system and the instantaneous vessel coordinate system.

1. Click *Navigation* in the Sensors list box so the selection is highlighted and the navigation data fields are visible.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current navigation time stamp.
   - *Time*: The hour and minute of the current navigation time stamp.
   - *Time Correction*: The time correction value.

The X-Y-Z fields set the location of the navigation source from the Reference Point (0).

- *X*: The athwart-ship distance of the source, positive to starboard.
- *Y*: The along-ship distance of the source, positive towards the bow.
- *Z*: The vertical distance of the source, positive into the water.
- *Ellipsoid*: Select an ellipsoid on which the navigation is based. The ellipsoid should be the same as the one used in the survey.
- *Manufacturer*: The maker of the navigation equipment.
- *Model*: The make of the navigation equipment.
- *Serial Number*: The serial number of the navigation equipment.

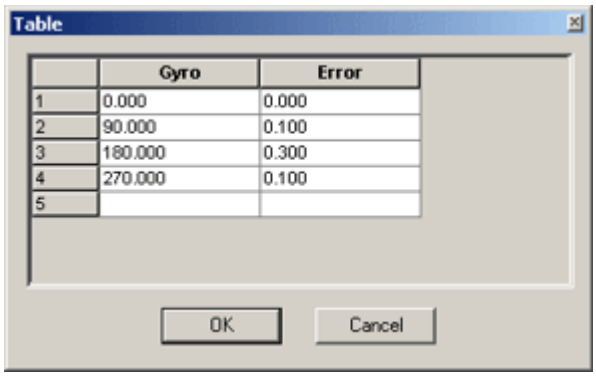
Gyro

The gyro sensor refers to any device providing a heading orientation of the vessel.

1. Click *Gyro* in the Sensors list box so the selection is highlighted and the gyro data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current gyro sensor time stamp.
   - *Time*: The hour and minute of the gyro sensor time stamp.
   - *Time Correction*: The time correction value.

The Gyro Error is measured in degrees and is the difference between the recorded sensor value and the applied heading of the vessel: `Applied Heading = Recorded value - Gyro Error`.

3. Click an **Edit** button in the *Error* column to open the Gyro Error Table.



4. Type new values in the *Gyro* or **Error** fields and click **OK** to close the dialog box.
5. Type data (as needed) in the following fields.
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The make of the sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

Heave

The heave sensor records the vertical motion of the vessel. Although the heave sensor has its own section in the Vessel Editor, it is usually part of the same sensor package as Pitch and Roll in most survey configurations.

1. Click *Heave* in the Sensors list box so the selection is highlighted and the data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current heave sensor time stamp.
   - *Time*: The hour and minute of the current heave sensor time stamp.
   - *Time Correction*: The time correction value.

The X-Y-Z fields set the location of the heave sensor from the Reference Position (0).

The X-Y-Z location of the heave sensor must be defined ONLY when you intend to apply heave data during Merge AND when remote heave compensation is necessary.

- *X*: The athwart-ship distance of the sensor, positive to starboard.
- *Y*: The along-ship distance of the sensor, positive to the bow.
- *Z*: The vertical distance of the sensor, positive into the water.
- Error: The applied instantaneous values for the heave sensor are computed by subtracting the appropriate errors from the recorded values:

  ```
  Applied Heave = Recorded Heave - Heave Error.
  ```

- *Apply*: Select Yes to apply heave data in the merge process.
- *Manufacturer*: The maker of the sensor equipment.
- *Model*: The make of the sensor equipment.
- *Serial Number*: The serial number of the sensor equipment.

The location of the heave sensor is shown in the Vessel Editor by a coloured dot.

Note: Simrad systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '"No" so these values are not applied twice.

Pitch

Pitch refers to the rotational motion of the vessel around the X (port/starboard) axis. Although the pitch sensor has its own section in the Vessel Editor, it is usually part of the same sensor package as roll and heave in most survey configurations.

1. Click *Pitch* in the Sensors list box so the selection is highlighted and the data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current pitch time stamp.
   - *Time*: The hour and minute of the current pitch time stamp.
   - *Time Correction*: The time correction value.
   - *Error*: The applied instantaneous values for the pitch sensor are computed by subtracting the appropriate errors from the recorded values:

     ```
     Applied Pitch = Recorded Pitch - Pitch Error.
     ```
   - *Apply*: Select Yes to apply the pitch data in the merge process.
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The make of the sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

The location of the pitch sensor is indicated in the Vessel Editor by a coloured dot.

Note: Simrad systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '"No" so these values are not applied twice.

Roll

Roll refers to the rotational motion of the vessel around the Y (fore/aft) axis. Although the roll sensor has its own section in the Vessel Editor, it is usually part of the same sensor package as pitch and heave in most survey configurations.

1. Click *Roll* in the Sensors list box so the selection is highlighted and the roll data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current roll time stamp.
   - *Time*: The hour and minute of the current roll time stamp.
   - *Time Correction*: The time correction value.
   - *Error*: The applied instantaneous values for the roll sensor are computed by subtracting the appropriate errors from the recorded values:

     ```
     Applied Roll = Recorded Roll - Roll Error.
     ```
   - *Apply*: Select Yes to apply the roll data in the merge process.
   - *Manufacturer*: The maker of the sensor equipment.
   - *Model*: The make of the sensor equipment.
   - *Serial Number*: The serial number of the sensor equipment.

The location of the roll sensor is indicated in the Vessel Editor by a coloured dot.

Note: Simrad systems apply dynamic Heave, Pitch, and Roll values to the swath data during survey. Therefore, the HVF must be set up with the *Apply* switches for Heave, Pitch, and Roll set to '"No" so these values are not applied twice.

Dynamic Draft

The squat and lift of a vessel changes as the speed changes. For some vessels, if the squat/lift is not accounted for, significant errors are introduced into the soundings. In this section, you can specify up to 10 speed-draft value pairs. During merge, the difference between the instantaneous draft and the static draft is computed and the final depth compensated.

Draft is measured in metres/feet, and speed is measured in knots. All draft values entered should be relative to the same reference, but what that reference is, is not important. The first Speed-Draft pair must correspond to a state where no draft correction is necessary. The delta draft values, relative to the initial draft, are computed and used to correct observed soundings.

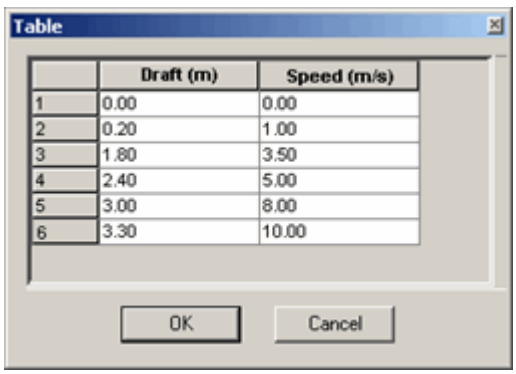
The following formula is used to correct soundings for dynamic draft:

```
Depth = observed depth – waterline + delta draft.
```

Delta draft is computed from the present vessel speed, as derived from the navigation system.

You can also create your own dynamic draft model and load Delta Draft information directly which will override the Dynamic Draft table in the HVF.

1. Click *Dynamic Draft* in the sensors list box so the selection is highlighted and the dynamic draft data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current time stamp for the draft table.
   - *Time*: The hour and minute of the current time stamp for the draft table.
   - *Apply*: Select Yes to apply the table during merge (or No to not apply the table).
3. Click the *Dynamic Draft* field. A dialog box containing a table with up to 10 speed-draft pairs is displayed.



4. Click inside the *Speed* cell and type a speed value (in knots).
5. Click inside the *Draft* cell and type a corresponding draft value.
6. Continue entering as many speed /draft pairs as needed.
7. Click **OK**.

The speed-draft table is saved in the HVF.

- *Comments*: Any additional comments.

Single Beam

The Single beam section refers to a vertical beam echo sounder with one transducer mount. It can be a single or dual frequency sounder defined as one beam with one transducer.

1. Click *Single Beam* in the Sensors list box so the selection is highlighted and the data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current single beam time stamp.
   - *Time*: The hour and minute of the current time stamp.
   - *Time Correction*: The time correction value.

The X-Y-Z fields set the location of the transducer from the Reference Point (0).

- *X*: The athwart-ship distance of the transducer, positive to starboard.
- *Y*: The along-ship distance of the transducer, positive to the bow.
- *Z*: The vertical distance of the transducer, positive into the water.
- *Manufacturer*: The maker of the transducer.
- *Model*: The make of the transducer.
- *Serial Number*: The serial number of the transducer.

SVP

A sound velocity profile (SVP) records the speed of sound at various depths in the water column. One of the processes available in HIPS is the application of SVP data in sound velocity corrections. However, this procedure is not available for all types of sonars because some sonars and their data logging systems compensate for sound velocity during data acquisition.

To ensure the SVP is accurately applied in HIPS, the transducer X-Y-Z offset values must be entered.

1. Click *SVP 1* in the Sensors list box so the selection is highlighted and the SVP transducer data fields are visible. If there is a second transducer then click *SVP 2* and complete the same procedures as listed below.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current SVP pole time stamp.
   - *Time*: The hour and minute of the current SVP pole time stamp.
3. Select Yes if there are dual transducers (or No if there is only a single transducer).

The following three fields are for entering the pole's X-Y-Z coordinates. All coordinates are measured from the Reference Point

4. Type the transducer's X-Y-Z offsets in the following fields:
   - *X*: The athwart-ship distance of the transducer, positive to starboard.
   - *Y*: The along-ship distance of the transducer, positive to the bow.
   - *Z*: The vertical distance of the transducer, positive into the water.

> Note: The Reference Point for Simrad data is the centre of rotation. It should be possible to retrieve offset values from the Simrad Installation Datagram.

The next three fields refer to the alignment of the transducer. The values you entered in the wizard when creating the HVF are displayed in these fields. The fields are only to be used for large transducer mounting offsets.

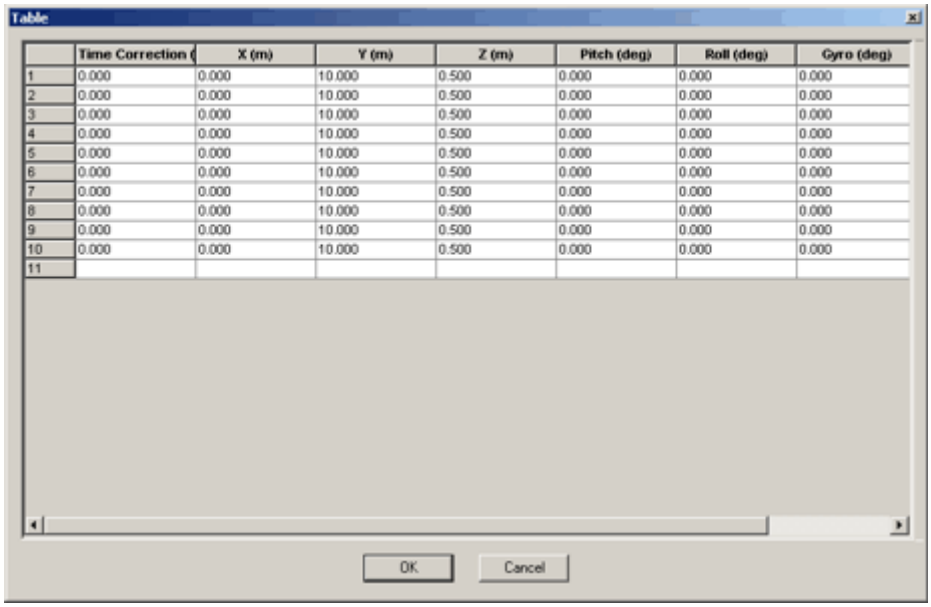
- *Pitch*: Pitch offsets of the transducer.
- *Roll*: Roll offsets of the transducer.
- *Yaw*: Rotation of the transducer (either 0 or 180 degrees).

For information on applying SVP in processing see "SOUND VELOCITY PROCESSING" ON PAGE 223.

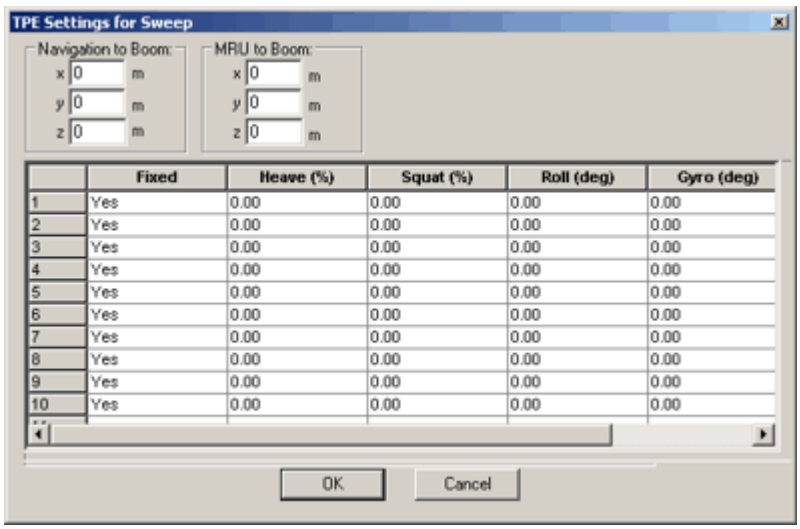
Sweep

Sweep systems typically contain multiple vertical beam transducers mounted on a boom and pointed straight down. The reference point for a sweep system can be anywhere on the X-Y plane, but must be on the water surface.

1. Click *Sweep* in the Sensors list box so the selection is highlighted and the data fields are visible.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current sweep time stamp.
   - *Time*: The hour and minute of the current sweep time stamp.
   - *Time Correction*: The time correction value.
   - *Beams*: Number of beams mounted on the boom.
3. Click the **Edit** button in the *Beam Status* field to view a table for entering data for individual beams on the boom.



4. Type values in the fields and click **OK**.
5. Click **Edit** in the *TPE Settings* field to enter TPE values for each beam on the boom and to edit values entered in the new vessel wizard.



6. Type values as needed and click **OK.**

Towed Sensors

If a towed sensor—such as a side scan sonar—is used on a survey, then SIPS can compute the sensor's position as from the ship's position by calculating the horizontal layback and a direction to the sensor.

1. Click *Towed* in the Sensors list box so the selection is highlighted and the Towed data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current towed sensor time stamp.
   - *Time*: The hour and minute of the current towed sensor time stamp.
   - *Time Correction*: The time correction value.
   - *Layback Error*: The error in the computed or recorded horizontal layback. This can be used, for example, to compensate the recorded layback data or tow cable length for the distance between the origin of the measurements and the defined tow point location in the tow cable length. This value is subtracted from the computed horizontal layback.
   - *X*: Offset of the tow point from the vessel's reference position.
   - *Y*: Offset of the tow point from the vessel's reference position.
   - *Z*: Height of the tow point (negative upwards) in relation to the datum that is referenced to the sensor depth (in most cases, this is the waterline).
   - *Manufacturer*: The maker of the towed sensor.
   - *Model*: The make of the towed sensor.
   - *Serial Number*: The serial number of the towed sensor.

Waterline

The purpose of this section is to define long-period changes in the vessel's draft due to fuel burn or other loading changes. Specifically, you can enter the height of the waterline below the Reference Point.

If the Waterline section of the vessel configuration is not defined then the default of zero is used by the software as the height of the waterline below the RP. The Waterline section must be present in the HVF for use in sound velocity corrections.

1. Click *Waterline* in the Sensors list box so the selection is highlighted and the waterline data fields are displayed.
2. Type data (as needed) in the following fields:
   - *Date*: The year and Julian day of the current waterline time stamp.
   - *Time*: The hour and minute of the current waterline time stamp.
   - *Waterline*: The distance from the RP, positive when below the RP.
   - *Apply*: Select Yes to apply the waterline data in the merge process.

For Simrad data, the Waterline value must be set to the same value recorded as **WLZ** in the Simrad Installation Datagram.

The Waterline value will only be used during Sound Velocity Correction.

The Apply switch must be set to **“No”**. It must not be set to “Yes”, because it must not be used by Merge.

Total Propagated Error

This section defines the values for calculating Total Propagated Error (TPE). TPE is derived from a combination of all individual error sources. The TPE Configuration section of the vessel file contains the following errors (among others):

- nav/gyro/heave/pitch/roll/tide errors
- latency error estimate
- sensor offset error estimates.

These errors are combined with individual sonar model characteristics in the DeviceModels.xml file to calculate horizontal and vertical uncertainty values for every sounding along a track line when TPE is applied.

1. Expand the TPE section by clicking + icon.
2. Type data (as needed) in the Offsets section:
   - *MRU to Transducer*: The physical offset in three dimensions from the motion recording unit to transducer 1 on the vessel.
   - *MRU to Transducer2*: The physical offset in three dimensions from the motion recording unit to transducer 2 on the vessel.
   - *Navigation To Transducer*: The physical offset in three dimensions from the navigation antenna to transducer 1 on the vessel.
   - *Navigation To Transducer2*: The physical offset in three dimensions from the navigation antenna to transducer 2 on the vessel.
   - *Transducer Roll*: The mounting roll offset for transducer 1. The offset is positive when rotating the transducer away from starboard (starboard down).
   - *Transducer Roll 2*: The mounting roll offset for transducer 2. The offset is positive when rotating the transducer away from starboard (starboard down).
3. Type data (as needed) in the Standard Deviation section:
   - *Motion Gyro*: The measurement standard deviation of the heading data in degrees.
   - *Heave % Amplitude*: An additional heave standard deviation component that is the percentage of the instantaneous heave.
   - *Heave (m)*: The measurement for standard deviation of the heave data. Most heave manufacturers quote heave error as being determined from StaticHeave or PercentageOfHeave depending on which value is larger.
   - *Roll*: The measurement standard deviation of the roll data in degrees.
   - *Pitch*: The measurement standard deviation of the pitch data in degrees.

- *Position Nav*: The standard deviation associated with the measurement of positions for the vessel. This is usually the error of the GPS sensor being used.
- *Timing Trans*: Standard deviation in transducer time stamp measurement.
- *Nav Timing*: Standard deviation in navigation time stamp measurement.
- *Gyro Timing*: Standard deviation in gyro time stamp measurement.
- *Heave Timing*: Standard deviation in heave time stamp measurement.
- *Pitch Timing*: Standard deviation in pitch time stamp measurement.
- *Roll Timing*: Standard deviation in roll time stamp measurement.
- *Offset X*: Standard deviation for the X measured offset on the vessel.
- *Offset Y*: Standard deviation for the Y measured offset on the vessel.
- *Offset Z*: Standard deviation for Z measured offset on the vessel.
- *Vessel Speed*: The standard deviation for the vessel speed measurements.
- *Loading*: Vertical changes during the survey because of fuel consumption, etc.
- *Draft*: The standard deviation in the vessel draft measurements.
- *Delta Draft*: The standard deviation in the dynamic vessel draft measurements.
- *MRU Align StdDev Gyro*: This value is the uncertainty of the motion recording unit placement within the vessel fixed coordinate frame.
- *MRU Align StdDev Roll/Pitch*: This value is the uncertainty of the motion recording unit placement within the vessel fixed coordinate frame.
- *Comments*: Any additional information.

6

Starting a New Project

Define Project

All projects are organized in a Project-Vessel-Day folder hierarchy. The default directory for projects is …\Hips\HDCS_Data, however you can modify this setting using the Options dialog box (see "DIRECTORIES" ON PAGE 97).

A project can be created with new Project-Vessel-Day folders, or new Vessel-Day folders can be added to an existing project.

All projects must contain a vessel file. Make sure that an appropriate vessel file has been created. See "DEFINE VESSEL FILE" ON PAGE 104).

If your raw data files are set up in a Project/Vessel/Day/Line structure and a HIPS Vessel File exists for that data, you can skip this step and use the Conversion wizard to create the HIPS directory structure. See "RAW DATA" ON PAGE 111 and "RENAME DAY AND LINE FOLDERS" ON PAGE 156 for more information.

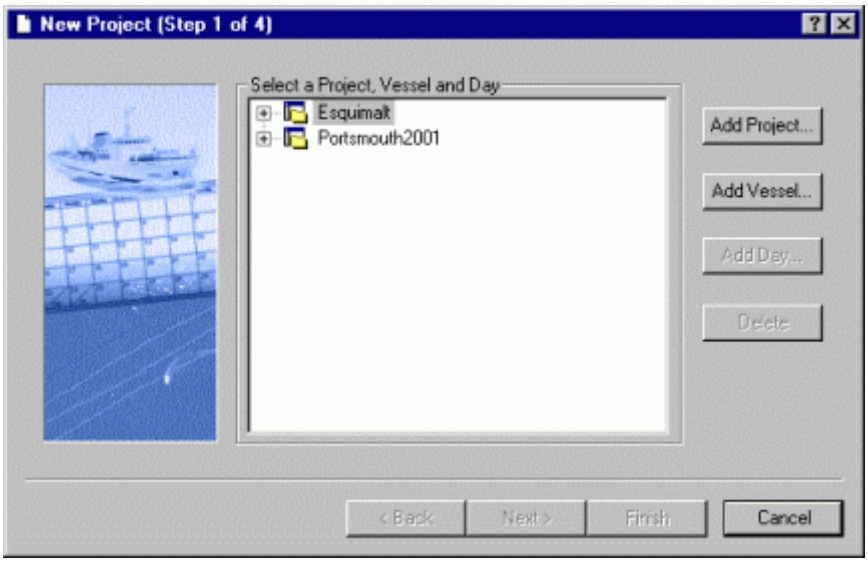
New projects are created through a wizard. Once all the data is entered into the wizard dialog boxes, a HIPS Project File (.hpf) is created.

1. Select the New Project command.

Menu	File > Project > New
Tool	

New Project (Step 1)

The New Project (Step 1) dialog box is displayed.



Add a new Project folder

1. To create a new project folder, click **Add Project**.

The New Project Name dialog box is displayed

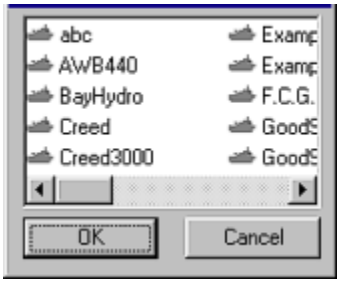
2. Type a name for the Project.
3. Click **OK**.

A new Project folder is created and displayed in the dialog box.

Add a new Vessel folder

The Vessel folder contains the Day folders for the project.

4. Select a Project folder to select a location for the Vessel folder.
5. Click **Add Vessel**.
6. The Add Vessel dialog box is displayed.



7. Select a vessel file.
8. Click **OK**.

The Vessel folder is created and added to the Project folder.

Add a new Day folder

The Day folder contains track line data after raw data has been converted to HIPS/SIPS format (see "RENAME DAY AND LINE FOLDERS" ON PAGE 156). The Day folder should be named with the date that the survey lines were recorded. However, for most formats, this is not necessary because the date is typically recorded with the raw data. In some cases the date of the Day folder will provide the date of the time reference in the HIPS/SIPS format.

9. Click a Vessel folder to select a location for the Day folder.
10. Click **Add Day**.

The Calendar is displayed at the current year-month-day.

11. Select a new year or month, if needed, by clicking the arrow buttons at the top of the dialog box.

12. Select a new day, if needed, by double-clicking a day button in the calendar dialog box.
13. Click **Add Day** again to create the Day folder with the selected dates. Day folders are shown in Julian Day format.
14. To delete an empty Day or Vessel folder, click **Delete**.

> The Delete button is disabled if a Vessel or Day folder already has data converted into it.

15. Click **Next**.

Select an existing project folder

New Vessel and Day folders can be added to an existing project.

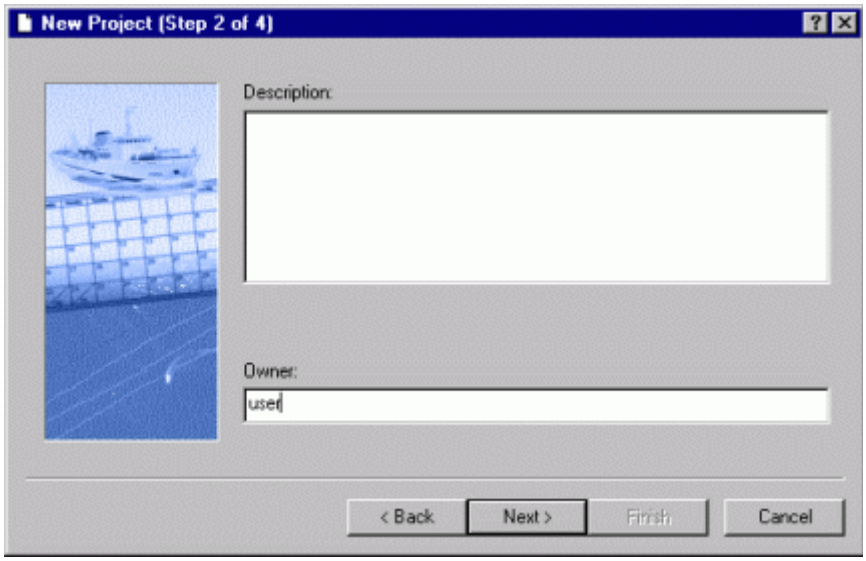
1. Click an existing project from the list in the dialog box.
2. Add vessel and day folders as need.

If a HIPS Project File (HPF) already exists for a project, the **Finish** button is displayed. You can enter a new Vessel or Day folder, but cannot enter information from the other wizard dialog boxes.

3. Click **Finish** to close the wizard.

New Project (Step 2)

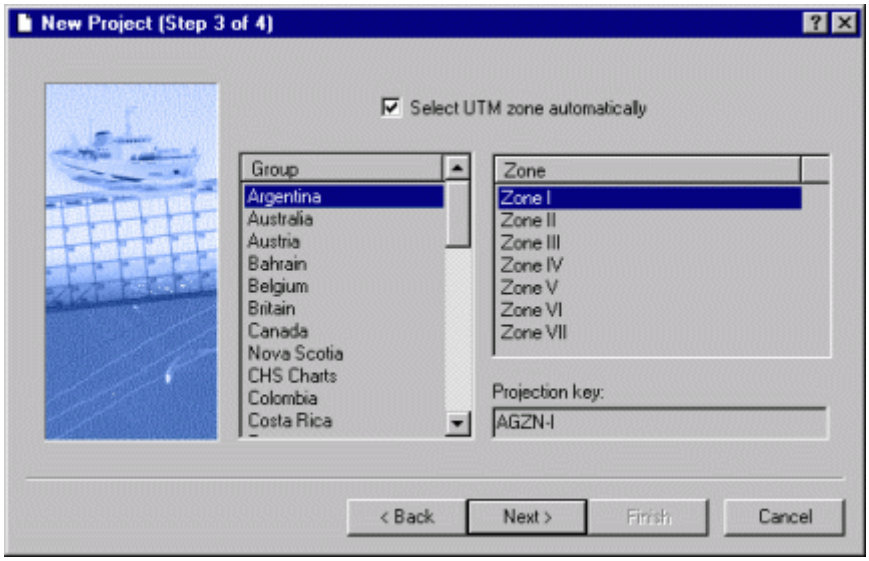
The New Project (Step 2) dialog box is displayed.



1. Type any comments about the project in the *Description* text area, if needed.
2. Enter the name of the person working with the project data in the *Owner* field. The default name that is displayed is taken from the current Windows NT logon name.
3. Click **Next**.

New Project (Step 3)

The New Project (Step 3) dialog box is displayed.



The world-wide map projection coordinate systems are defined in the file …\Hips\System\mapdef.dat.

1. Select the *Select UTM Zone Automatically* check box if you want to apply this projection. UTM is the default projection for HIPS and SIPS.

The country and zone options are dimmed.

If you are working with geographic projections, complete Steps 2 to 4.

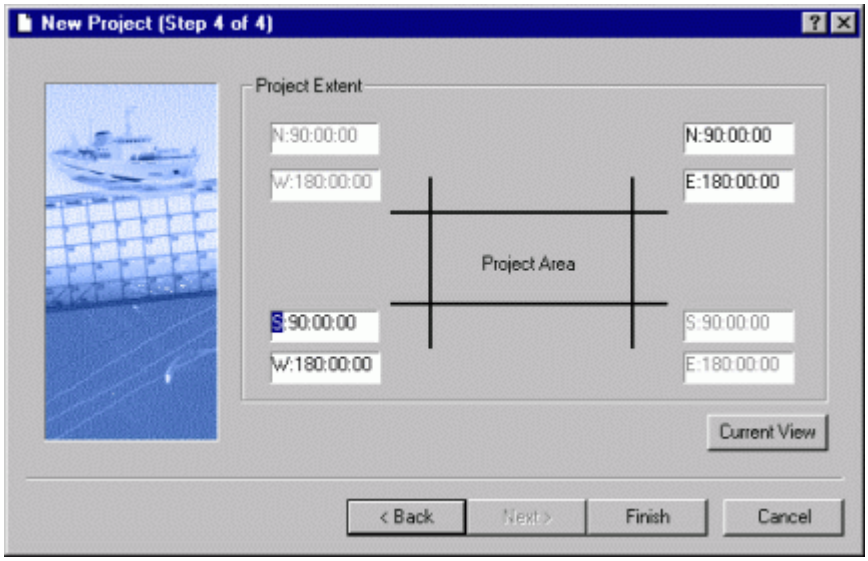
2. Click the UTM option so the box is no longer checked.
3. Select a country or area from the *Group Name* list box.
4. Select a zone from the *Entry Name* list box.

The key name is automatically displayed once the country and zone have been selected.

5. Click **Next**.

New Project (Step 4)

The New Project (Step 4) dialog box is displayed.



1. Enter the geographic coordinates for the project area (degrees-minutes-seconds) by clicking a section in a box and typing the information, or using the arrow keys to toggle the values into the entry fields.
2. If you want to use the geographic coordinates of a currently open project, click **Current View**.
3. Click **Finish**.

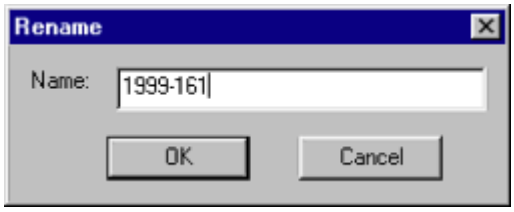
New Project-Vessel-Day folders have been created according to the options selected in the wizard. The HPF is now located in the Project folder.

Rename Day and Line Folders

You can rename existing Day and Line folders.

1. Select a Day folder or Line folder in the Control window, or select a track line in the Display window.
2. Select the Rename command.

Menu	Edit > Rename
Pop-up	Rename



The Rename dialog box is opened with the selected file or folder name displayed.

3. Enter a new name in the box.
4. Click **OK**.

The changed file name is displayed in the Control window.

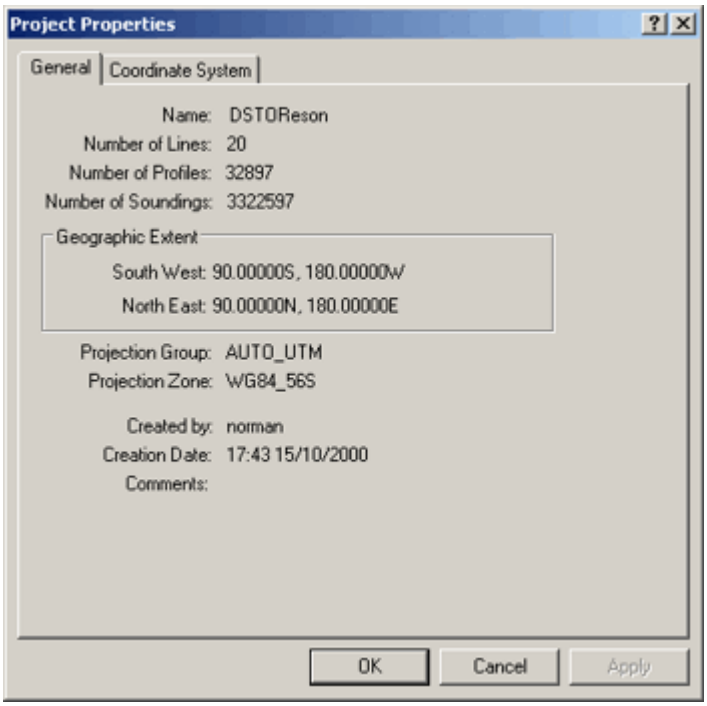
View Project or Vessel Properties

View essential project and vessel information. You can also use this dialog box to change the coordinate system, if needed.

1. Select a Project or Vessel folder from the Project tab in the Control window.
2. Select the Properties command.

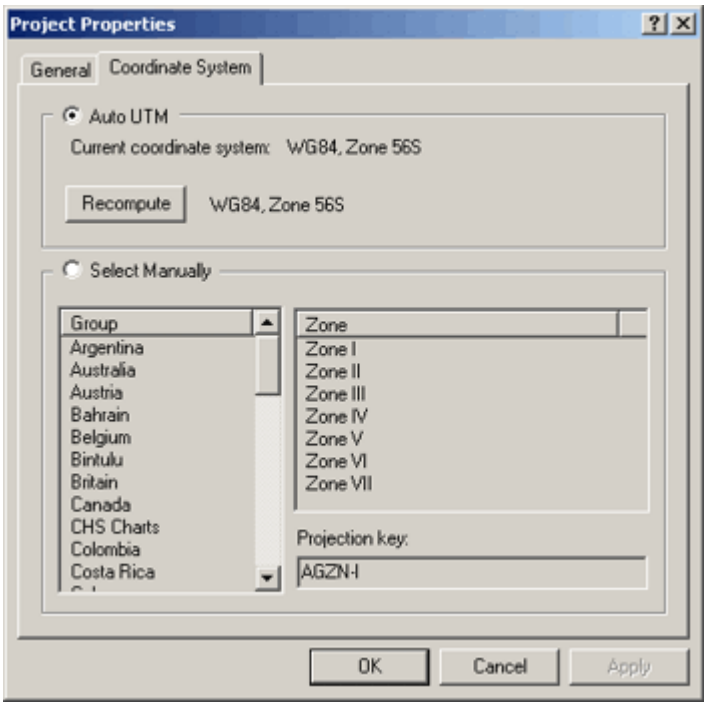
Menu	View > Properties
Pop-up	Properties

The Properties dialog box is displayed.



The General tab displays information about the project or vessel. The information is read-only and cannot be modified.

If you selected to display project properties, then the a Coordinate System tab is included in the dialog box.



You can select another coordinate system for the project using this tab.

3. Select either the *Auto UTM* option or the *Select Manually* option.
4. If you selected Auto UTM, click Recompute to compute the UTM zone.
5. If you want to manually select a coordinate system, select a projection and zone.
6. Click **Apply** to assign the new coordinate system to the project.
7. Click **OK** to close the dialog box.

Session Files

Session files are used for re-opening all data that was processed the last time the project was open. The data types that are recorded in the session file include:

- lines
- field sheets
- background data
- last geographic view extents of the Display window

All session files have an .hsf file extension and are stored by default in …\Hips\Session. The previous format of the session file (.ses) can be opened and automatically upgraded to the new session file format.

Save a session

Menu	File > Save Session/ Save Session As

1. Select the Save Session command, or select the Save Session As command if the session has not been saved before, or if you want to save an existing session under a new name.
   - If this is an already saved file, the program re-saves the existing .hsf file.
   - If this is a new file, then the Save As dialog box is displayed. Select a folder where you want to save the .hsf file and type a name in the File Name text box.

If a project contains a large number of track lines, you can make data processing more manageable and reduce the time it takes to load data into the application by selecting specific track lines for a session file.

1. Select the track lines you want to save in the session file.
2. Choose the Save Session As command and select the *Save Selected Lines Only* check box.
3. Type a name for the session file and click **Save**.

The Session file is saved.

Open a session

Menu	File > Open Session

1. Select the Open Session command.

The Open Session File dialog box is displayed.

2. Select a session file.
3. Click **Open**.

The data listed in the Session file is opened and the layers are shown in the Session tab in the Control window.

Close a session

Menu	File > Close Session

1. Select the Close Session command.
2. If you have made changes to the session since the last save, you are prompted to save the current changes.
3. Click **OK**.

Session file shortcut

The session file extension (.hsf) is a registered Windows extension. Therefore, you can create a shortcut on the desktop for the session and then double-click the shortcut to start the Hips application and automatically reopen the session file.

Add Session Files

While a session file is open, you can display data from another session file in the already open one.

Menu	File > Add Session

1. Select the command.

The standard Open dialog box is displayed.

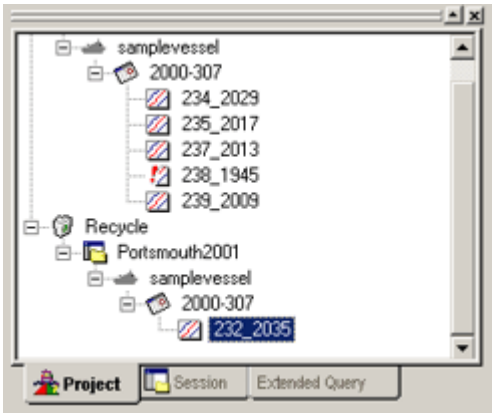
2. Select a directory and file.
3. Click Open.

The file is opened in the application.

Recycle Bin

You can “delete” Project, Vessel, Day, or Line folders from an active project by moving them to the Recycle Bin. These folders are no longer part of the active project, but can be restored to the project or permanently deleted. All folders in the Recycle Bin are located in …\Hips\Recycler.

The Recycle Bin recreates the same directory structure as the active project.



Move items

Menu	Window > Control
Tool	

1. Ensure the Control window is open and the Project tab is visible.
2. Select the folder(s) you want to delete.

Menu	Edit > Delete
Pop-up	Delete
Key	<Delete>

3. Select the Delete command.
4. A dialog box asks if you want to delete the folder(s).
5. Click **Yes**.
6. [Optional] You can also select the folders and, while pressing the mouse button, drag them to the Recycle Bin.

The folders are moved to the Recycle bin.

Restore an item

You can move a previously deleted folder from the Recycle Bin to the active project.

1. Select the folders in the Recycle Bin.

2. Select the Restore command.

Menu	Edit > Restore
Pop-up	Restore

The folders are now restored to the active project. If all folders are removed from the Recycle Bin then the directory structure is deleted.

Empty Recycle Bin

Emptying the Recycle Bin permanently deletes the folders from the project.

> You cannot restore folders that have been deleted from the Recycle Bin.

3. Select the Empty Recycle Bin command.
4. A dialog box asks you if you want to continue.
5. Click **Yes**.

Menu	Edit > Empty Recycle Bin
Pop-up	Empty Recycle Bin

The folders are removed from the Recycle Bin.

Archive Project

Archive projects and related data using the Project Archive wizard.

The projects and data are converted to a ZIP format and must be decompressed if used again. The directory structures for the compressed files are as follows:

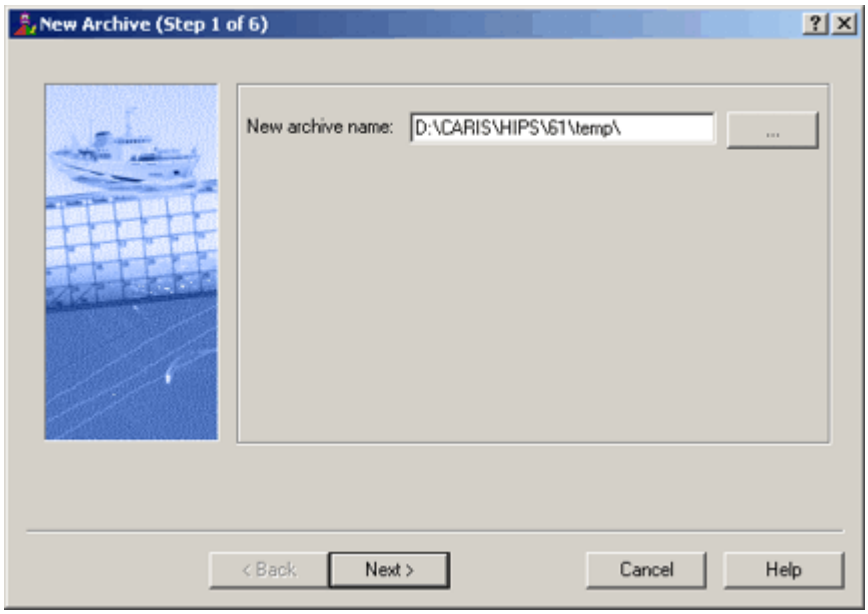
- Line files are in ../Projects/<ProjectName>/<VesselName>/<DayName>/<LineName>
- Fieldsheet files are in ../FieldSheets/<ProjectName>/<FieldSheetName>
- Other files related to data processing are in ../Preprocess/<ProjectName>
- Additional background files are in .. /Background/<ProjectName>

To archive a project.

1. Select the Archive Project command.

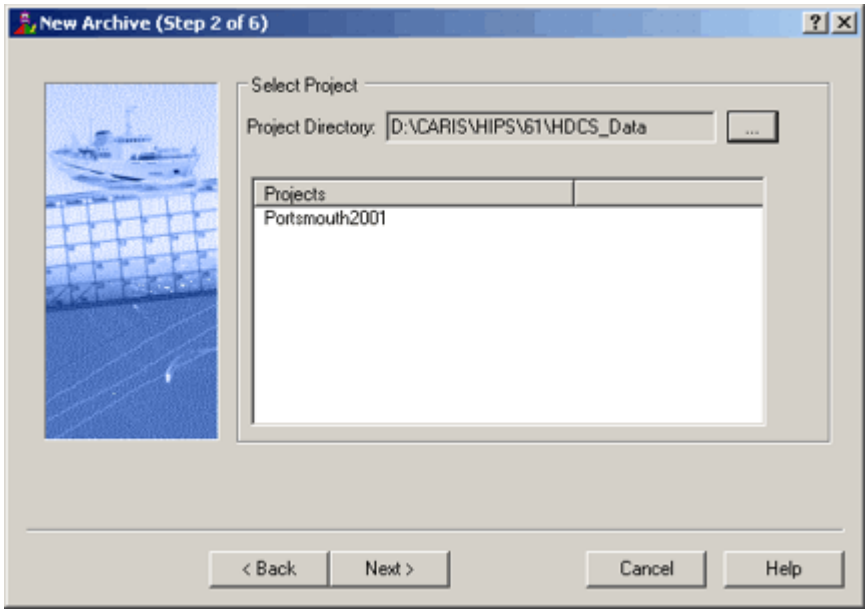
Menu	File > Project > Archive

The New Archive (Step 1) dialog box is displayed.



2. Type the file path for the new archived file in the New Archive Name field; or click Browse to select a directory and type the name of the archived project file.
3. Click **Next**.

The New Archive (Step 2) dialog box is displayed.

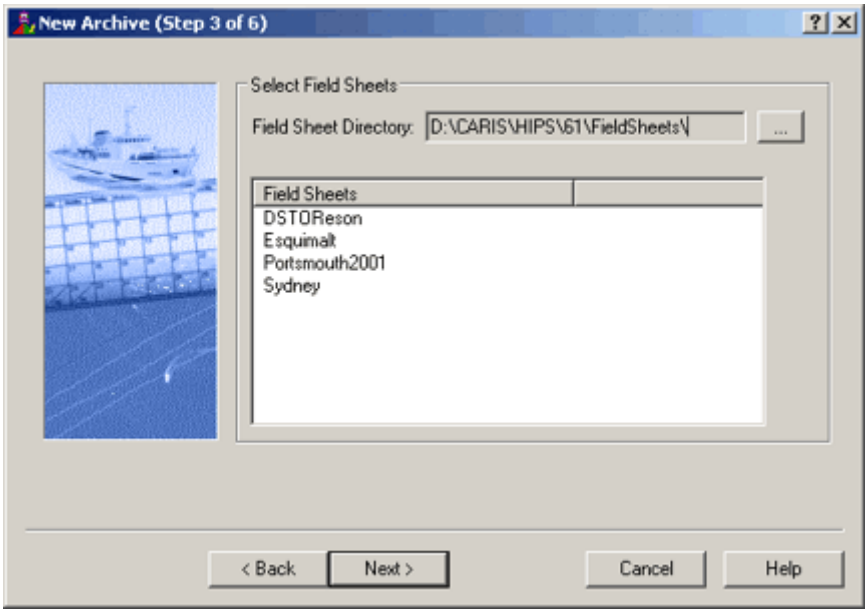


4. The directory path for the projects is shown in the Project Directory field. If necessary, click Browse and select another directory.

The projects in that directory are displayed in the *Projects* list.

5. Select a project n the list.
6. Click **Next**.

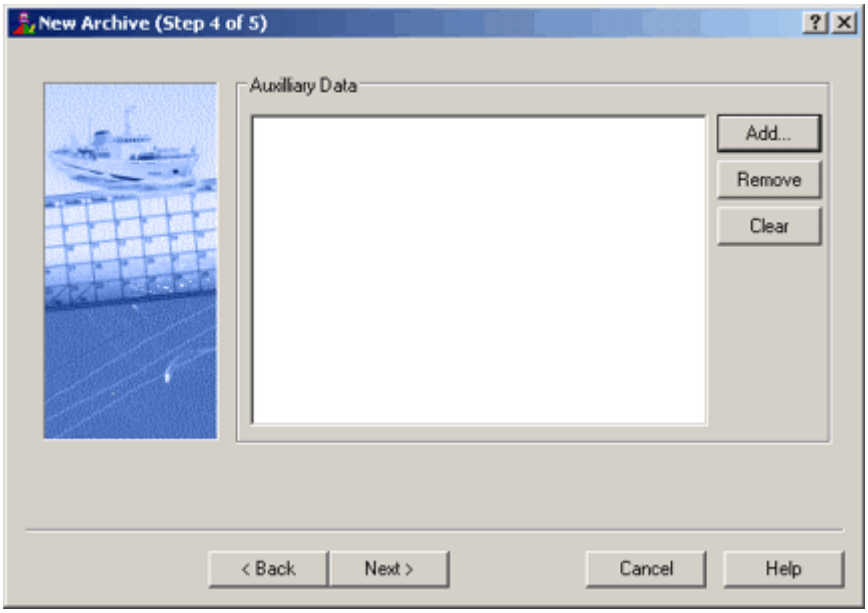
The New Archive (Step 3) dialog box is displayed.



7. The directory path for field sheets is shown in the Field Sheet Directory field. If necessary, click Browse to select another directory.

The field sheets contained in the directory are displayed in the Field Sheet list.

8. Select a field sheet associated with the project using three methods:
   - Select a single field sheet by clicking on it.
   - Select a group of field sheets by pressing and holding the <Ctrl> key while clicking on each field sheet.
   - Select a range of field sheets by pressing and holding the <Shift> key while clicking on the first and last field sheets in the range so that all field sheets in between are highlighted.
9. Click **Next**.
   - The New Archive (Step 4) dialog box is displayed.



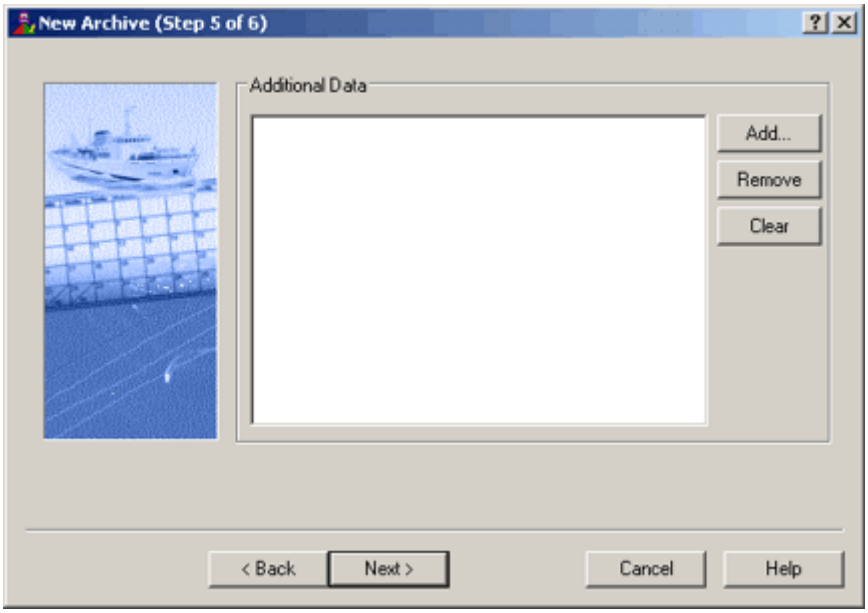
This dialog box is used for archiving related files necessary for the successful processing of data. Examples of these files would be tide files, sound velocity files, delta draft, TrueHeave files, etc.

10. Click **Add** to view a standard Open dialog to select auxiliary data.

The auxiliary files are shown in the *Auxiliary Data* list.

11. To remove a file, select it and click **Remove**, or click **Clear** to remove all listed files.
12. Click **Next**.

The New Archive (Step 5) dialog box is displayed.



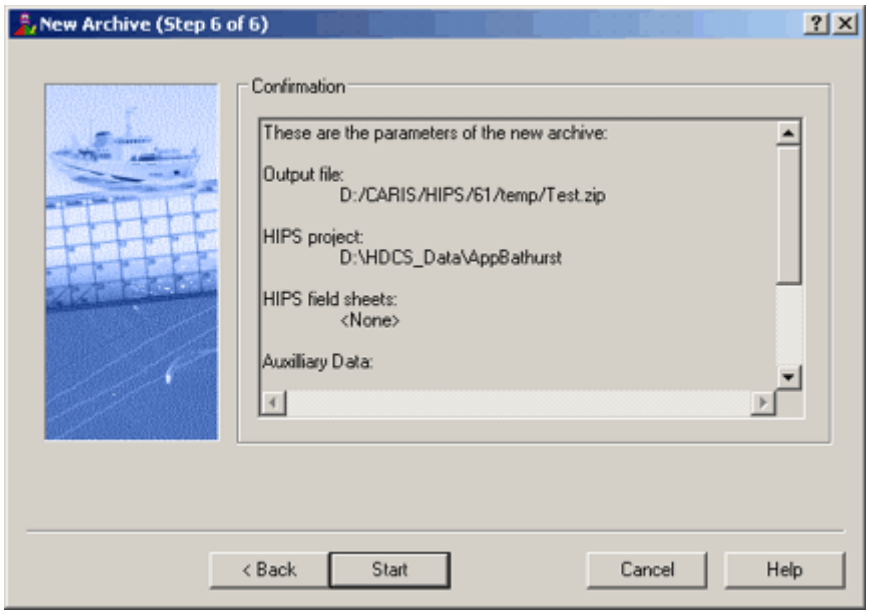
This dialog box is used for archiving data that is associated with the project, but is not necessary for processing data. Examples of this type of data would be background images, associated charts, etc.

13. Click **Add** to view a standard Open dialog to select auxiliary data.

The auxiliary files are shown in the *Auxiliary Data* list.

14. To remove a file, select it and click **Remove**, or click **Clear** to remove all listed files.
15. Click **Next**.

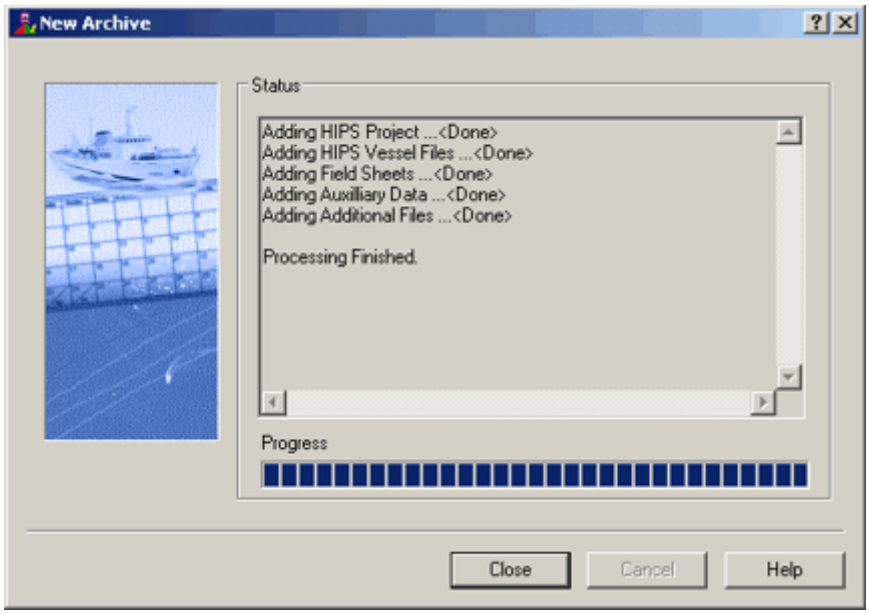
The New Archive (Step 6)dialog box is displayed.



This dialog box lists the selections you made in the previous dialog boxes.

16. Click **Back** to go to a previous dialog box to make changes.
17. Click **Start** to begin the archiving process.

The New Archive (Final) dialog box is displayed.



The Status list displays the files that have archived. A progress bar displays the amount of archiving that has been accomplished.

18. Click Close to exit the wizard once archiving is completed

7

File Conversion

Converting Files to HIPS/SIPS Format

HIPS and SIPS files are created from survey data using the Conversion Wizard.

Data that is converted to HIPS/SIPS format may be in varying stages of completion, depending on the data format options. Survey data may or may not have been corrected for factors such as heave/pitch/roll or sound velocity.

All HIPS and SIPS data is organized in a Project/Vessel/Day/Line directory structure. (see "DEFINE PROJECT" ON PAGE 150).

If you organize the raw data files according to this structure, then you can convert line data from entire Project or from multiple Vessel and Day folders contained in a common Project folder. Use the File Selection Type option in Step 2 of the wizard to do this (see "SELECT FILES FOR CONVERSION" ON PAGE 172).

When entire project data has been converted, a Project-Vessel-Day-Line directory is created for the converted data. If there is already a Project folder that is identical to the preprocess folder, then the line data in the existing folder is over-written with the new data.

> You must organize your raw data files in a HIPS/SIPS Project-Vessel-Day structure to convert complete Project, Vessel and/or Day data files.

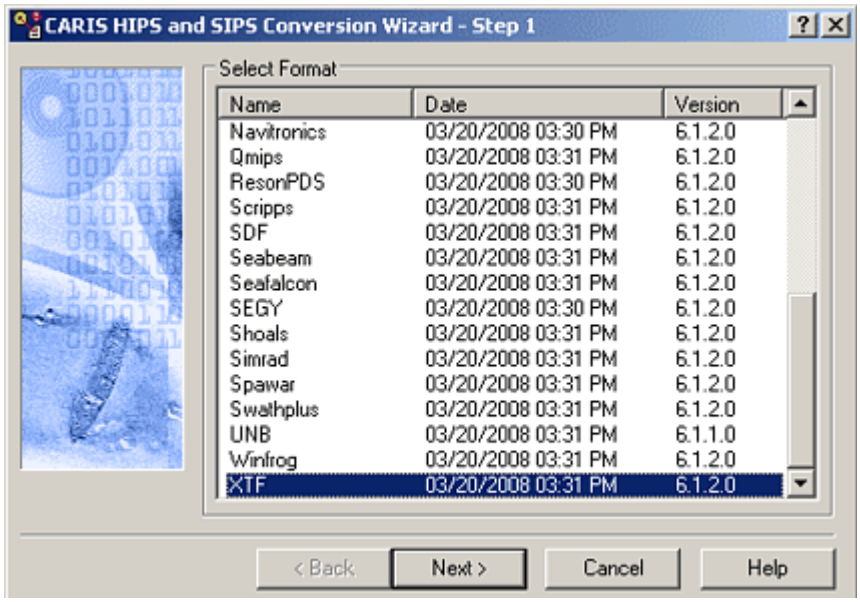
To start the conversion process, activate the HIPS Conversion Wizard:

1. Select the Conversion Wizard command in the HIPS and SIPS main interface.

The first Conversion dialog box is displayed.

Menu	File >Import > Conversion Wizard
Tool	

Select Data Format



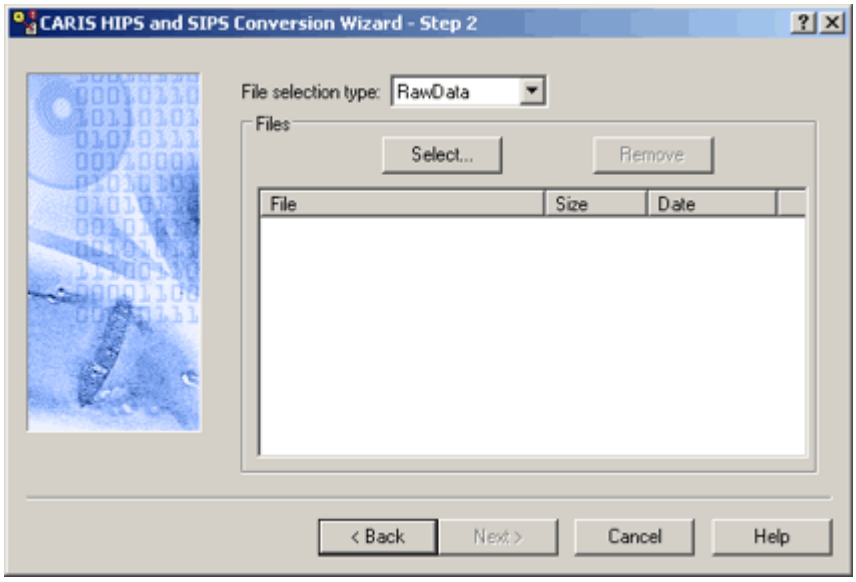
The Step 1 dialog box lists the data acquisition systems for which data can be converted to HIPS and SIPS files.

This list is created dynamically by searching the Hips\Bin directory for all format libraries with the name `convert_*.dll`.

1. Select the survey data type to be converted by clicking the appropriate name.
2. Click **Next**.

The next dialog box is displayed to select the files for conversion.

Select Files for Conversion



Raw data being converted into HIPS and SIPS can also be located on a read-only media such as a CD-ROM, except for Atlas SURF data, which at this time, does not support conversion from read-only files.

To convert data directly to a new or existing project directory,

1. Select the appropriate File Selection Type from the drop-down list:
   - Project to convert all files in the Vessel and Day folders
   - Vessel to convert all files in the Day folders
   - Day to convert all line files in a specific Day folders
   - Raw Data to convert individual line files in a Day folder.
2. Click **Select** to choose the files you want to convert.

The Select Files dialog box is displayed. The default directory is …\Hips\Preprocess, or you can browse to locate the files you want to convert.

3. Select the Project/Vessel/Day or raw data files and click **Open**.

The selected file are listed in the dialog box.

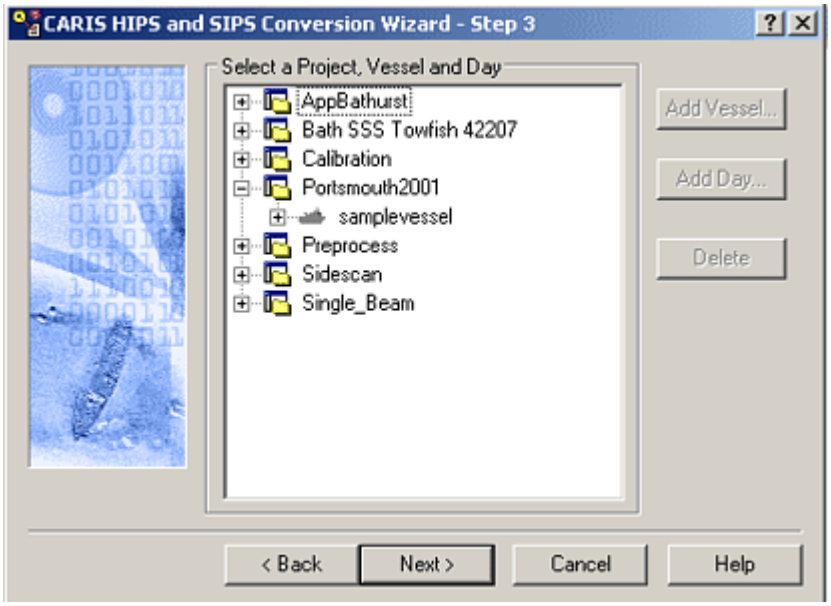
4. [Optional] To remove a file, select it and click **Remove**.
5. Click **Next**.

If you selected Project as the File Selection Type, then a Project-Vessel/Day/Line directory that is identical to the preprocess directory is created, and the wizard displays the Select

Coordinate Type dialog box (see "SELECT COORDINATE SYSTEM" ON PAGE 174).

Otherwise, the Select Project-Vessel-Day dialog box (Step 3 of the Conversion Wizard) is displayed.

Select Project-Vessel-Day



Select a Project/Vessel/Day directory to save the converted data.

1. Click the Expand (+) icon on the Project file tree so it is expanded and the Day folder is visible.

If you want to add Vessel or Day folders or to delete a folder, continue with Steps 2 to 6 in this procedure, otherwise go to Step 7.

2. To add a vessel folder, make sure the project directory is highlighted and click **Add Vessel**.

The Available Vessels dialog box is displayed.

3. Select a vessel folder and click **OK**.

The new Vessel folder is displayed in the selected project directory.

4. To add a day folder, click the Vessel folder and click **Add Day**.

The Calendar dialog box is displayed.

5. Select a year/month/day from the calendar, and click **OK**.

A Day folder with the selected date is stored in the Vessel folder.

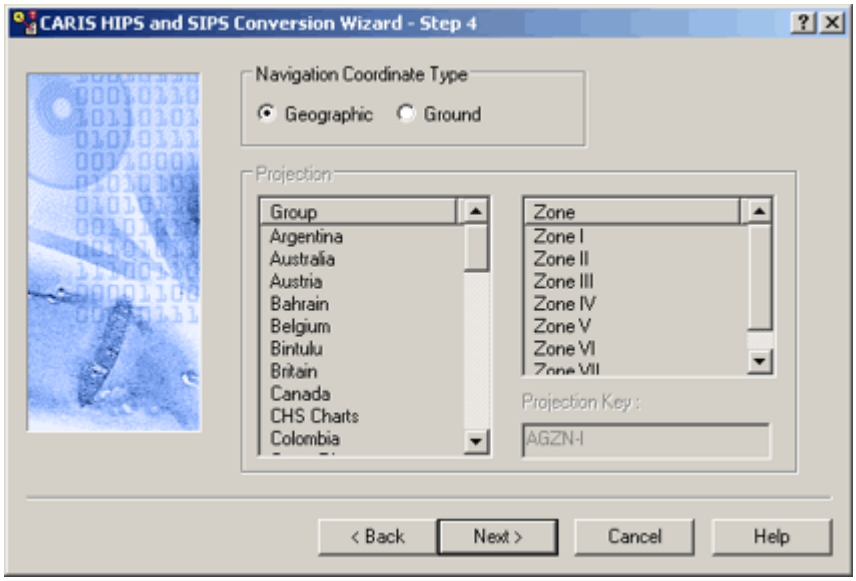
6. To remove an empty Day or Vessel folder from the wizard, highlight the folder and click **Delete**.

   If a Vessel or Day folder is in a Project which already has data converted into it, it cannot be deleted using the **Delete** button.

7. Click the Day folder so it is highlighted.
8. Click **Next**.

Select Coordinate System

The next step is to define the type of coordinates that were used to record the navigation data in the raw data files.



To indicate the type of coordinate system that was used for navigation data during data logging,

1. Select one of the following options:
   - *Geographic*: navigation data is recorded as latitude and longitude coordinates.
   - *Ground*: navigation data is recorded as eastings and northings.

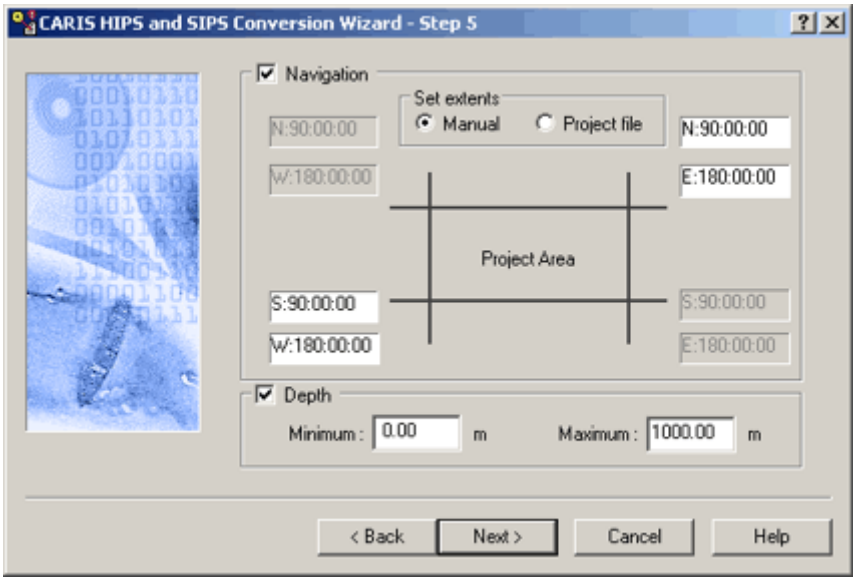
If you select *Ground*, complete the following steps:

2. Select an area from the *Group* list.
3. Select a zone.
4. Click **Next**.

Select Basic Filters

This dialog box defines basic filters to automatically reject extremely large errors in the recorded navigation and depth data.

During the conversion process, some soundings may be automatically rejected due to disabled beams. This is controlled by the beam status settings in the vessel configuration. Also, several multibeam formats pre-flag some soundings as being failed detects. These soundings are also automatically rejected during conversion.



Navigation and depth filters can be applied during conversion with the following steps:

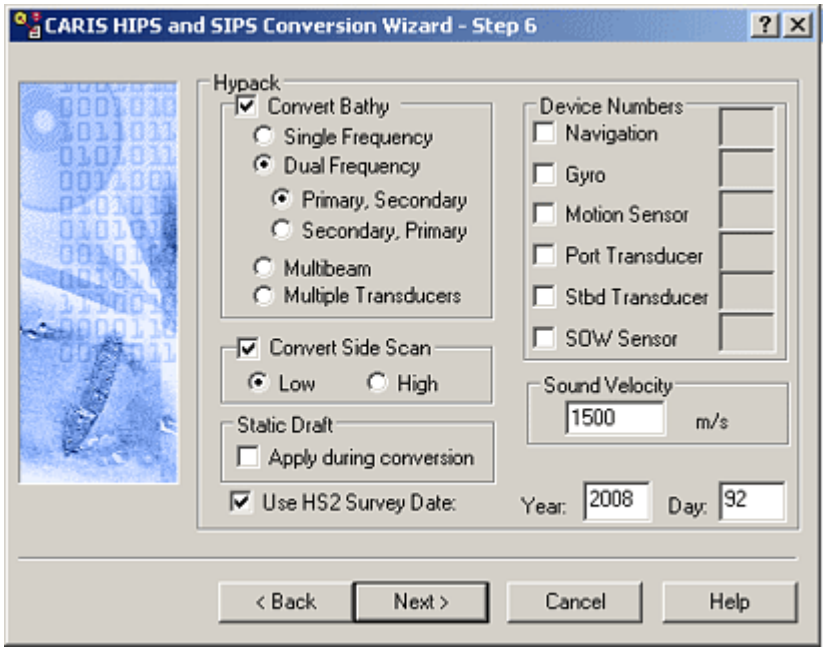
1. Select the *Navigation* check box to enable the navigation filter.

The extents of the filter can be defined either by setting them manually in the dialog box or by using the extents in the HIPS Project File (HPF). The HPF is from the project you selected in Step 3 of the Conversion wizard.

2. Select either *Manual* option or the *Project File* option.
3. If you selected the *Manual* option, enter the extents in the fields. You can use the <Tab> key to move between fields and use the arrow keys to move between values in a field.
4. Select the *Depth* check box to enable the depth filter.
5. Enter the minimum acceptable depth.
6. Enter a maximum acceptable depth.
7. Click **Next**.

Enter Data Parameters

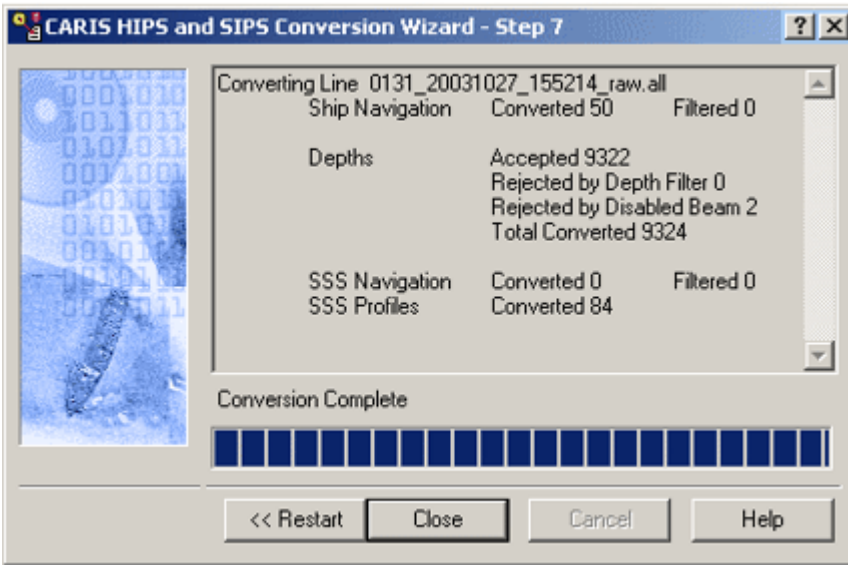
Depending on the data acquisition format you selected in the Step 1 of the wizard, one or more dialog boxes, appropriate to the data, will be displayed. For example, the image below shows the information for conversion of Hypack data.



1. Choose the settings appropriate to the data format being converted. See “DATA FORMAT OPTIONS” ON PAGE 178 for more information on options for different raw data formats.
2. Click **Next**.

Convert Data

The final step is to launch the conversion process with the settings and parameters you have selected.



To convert raw data to HIPS/SIPS format,

1. Click the **Convert** button.

A progress indicator bar along the bottom of the box is activated as the files are converted. When the operation is complete the results of the conversion process are displayed in the dialog box.

The converted files are saved in the directory you selected in Step 2 of the conversion process (see "SELECT FILES FOR CONVERSION" ON PAGE 172).

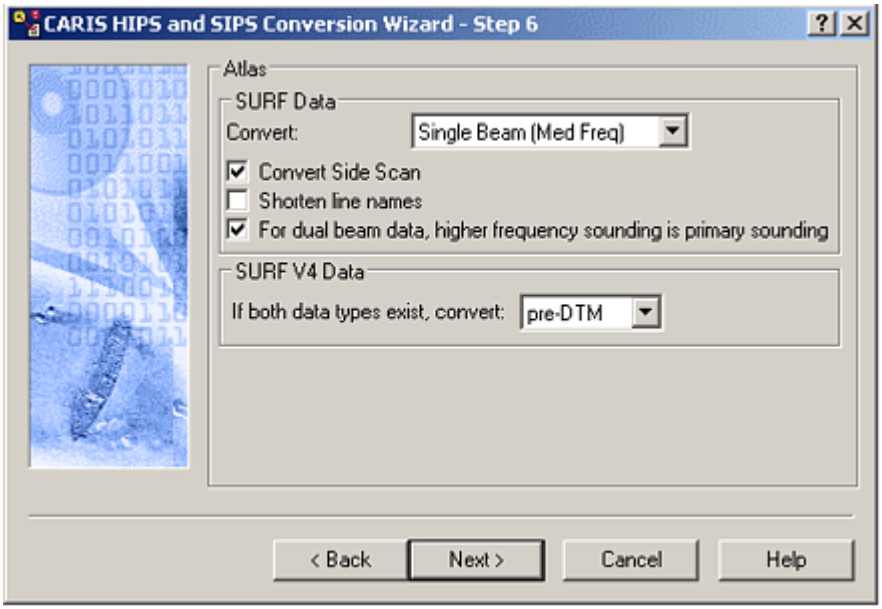
2. Click **Close** to close the wizard.
3. [Optional] to return to the initial step of the conversion process click **Restart.**

Data Format Options

Beginning with Step 6 of the HIPS and SIPS Conversion Wizard, one or more dialog boxes with specific format options is displayed. The number of options depends on the raw data format that was selected in Step 1 of the wizard.

The various data acquisition formats and their conversion options are presented below.

Atlas



1. Select one of the following SURF data types from the pull-down menu:
   - multibeam
   - single beam (low frequency)
   - single beam (medium frequency)
   - single beam (high frequency)
   - dual frequency (low-medium)
   - dual frequency (low-high)
   - dual frequency (medium-high)
2. Click the *Convert Side Scan* check box to include side scan data in the conversion process.

Critical line name information can be fitted into 12 characters provided by the CARIS Source ID attribute for soundings in a CARIS map.

3. Select the *Shorten line names* check box to modify the file name to a 12-character name (YYDDD_HHMMDD).
4. Select the higher frequency data as the primary soundings (if data format is dual-frequency).

At the time of this writing, the Atlas SURF library cannot be converted from read-only files. Therefore, the Atlas files must be located on a read-write media before they can be converted into HIPS.

Chirpscan3D

There are no options for the Chirpscan3D format.

CMAX

There are no options for the CMAX format.

Coda

1. Select a sonar channel.
2. Click the *Decimate Imagery* check box to apply a decimation routine to the side scan data. Decimation reduces side scan data to 1024 or fewer intensities per side for each swath.
3. Click the *Correct Imagery* check box to increase contrast in the side scan imagery.
4. Select the *Cable Out is Horizontal Layback* option to omit sensor depth during conversion. If there is no sensor depth and the towpoint has a Z-offset of zero, the cable out value is treated as the horizontal layback value during processing.

EdgeTech

1. Select a side scan system:
   - Midas
   - 260
   - JSF
2. If Midas is enabled, select a sonar frequency channel:
   - 100 KHz
   - 500 KHz
3. If JSF is enabled, select a sonar frequency channel:
   - All data
   - Low Frequency
   - High Frequency

Edgetech side scan data can be imported in 16-bit format or converted to an 8-bit format.

4. Select either the *Preserve 16-bit* option or *Convert to 8-bit* option.

If the *Convert to 8-bit option* is selected, the *Scale* and *Shift* options are enabled. The *Scale* option averages the data and the *Shift* option selects a initial bit value (between 0 and 8) and includes the next seven bit values for export. For example, if you typed 8 as your initial value, then bits 8 to 15 are included in the export process.

5. Select the *Scale* option to average the data into 8-bit format, or select *Shift* and type the value for the first bit.

EIVA

There are no options for this format.

Elac

During conversion, XSE-format beams are numbered according to the sounding's position along the swath (in relation to the vessel).

1. To number the converted beams along the swath from starboard to port, select the *Starboard to Port* check box.
2. To number the converted beams along the swath from port to starboard, select the *Port to Starboard* check box.

If a single Sound Velocity value was used during the survey:

3. Select the *Manual* option and type the sound velocity value.

If a Sound Velocity Profile was recorded and used by the logging system, then the *Interpolate* option must be selected.

4. Select *Interpolate* to read surface sound velocity values from a datagram.
5. To convert side scan data, select the *Convert Side Scan* check box.

ELAC XSE data can be recorded as 108 beams or 126 beams (depending on the survey mode). The *Pad Transducer 1 with NULL beams* writes 63 beams to Transducer 1 so that the same HIPS Vessel File (HVF) can be used for both survey modes.

6. Click the *Pad Transducer 1 with NULL beams* check box so that the soundings associated with transducer 1 are padded to have 63 beams.
7. Click **Next**.

Furuno

1. Make sure the *Convert Side Scan* option is checked to include side scan data with the bathymetric data.

GeoAcoustics

If you selected the Geographic in Step 4 of the wizard, you can obtain the latitude/longitude readings from GPS strings.

1. Select *Any GPS String* or *GGK*, *GGA*, or *GGL*.

The *Amplitude Filtering* option filters soundings according to amplitude value. For each ping (port and starboard pings are handled separately), the min/max amplitude values are obtained, and samples that fall below the selected percentage (0-50%) are rejected.

2. Select the *Amplitude Filtering* check box to implement this option.
3. Click the up or down arrow buttons to select a percentage for rejecting soundings.

The *Range Filtering* option converts soundings within a specified distance. Soundings outside of this range are not converted.

4. Select the *Range Filtering* check box to implement this option.
5. Click the up or down arrow buttons to select a minimum and maximum distance.

The *Pre-Filtering* option controls which soundings in each swath are considered for conversion. This option calculates the mean depth and standard deviation within a swath sector or horizontal bin. It then prevents any soundings that fall outside a multiple of the standard deviation from being converted.

If Pre-filtering is used, then the mean within the sector is re-computed. Ultimately, the soundings are sorted by their residual from the mean and those closest to the mean are converted. The actual number of soundings converted from within each sector is determined by the thinning factor (if used).

6. Select the *Pre-Filtering* check box to implement this option.
7. Determine a *Threshold* (multiples of the standard deviation) value by clicking the up and down arrow buttons.

The equivalent confidence value is displayed as a percentage.

The filtering and thinning methods are executed within a sector angle interval or a horizontal bin size. The sector angle option divides the swath into sectors according to degree angles while the horizontal bin size divides the swath into horizontal sectors based on a specified across-track distance.

Vessel motion and transducer mounting angles are considered when sector angle and horizontal bin locations are determined.

1. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
2. As an alternate method, Select the *Horizontal Bin Size* option choose an across-track distance by clicking the up or down arrow buttons.

The thinning factor reduces the number of soundings converted on a per swath basis. The thinning factor uses a power of two reduction control (1/2, 1/4, 1/8, etc.) so that one of two, one of four or one of eight soundings can optionally be converted per swath sector.

3. Select the *Data Thinning* check box to implement the thinning option.
4. Select a *Thinning Factor* value by clicking the up or down arrow buttons.
5. Click **Next**.

GSF

Data recorded during line turns it is marked as Offline data in GSF. This data can be included or excluded during conversion. When included it can also be automatically rejected. One purpose for including the Offline data would be to maintain the same swath profile numbers in HIPS format.

1. Make sure the *Include Offline Data* box is checked to convert data recorded during turns between track lines. Otherwise select the *Reject Offline Data* check box.

When offline data is converted, it can be rejected automatically.

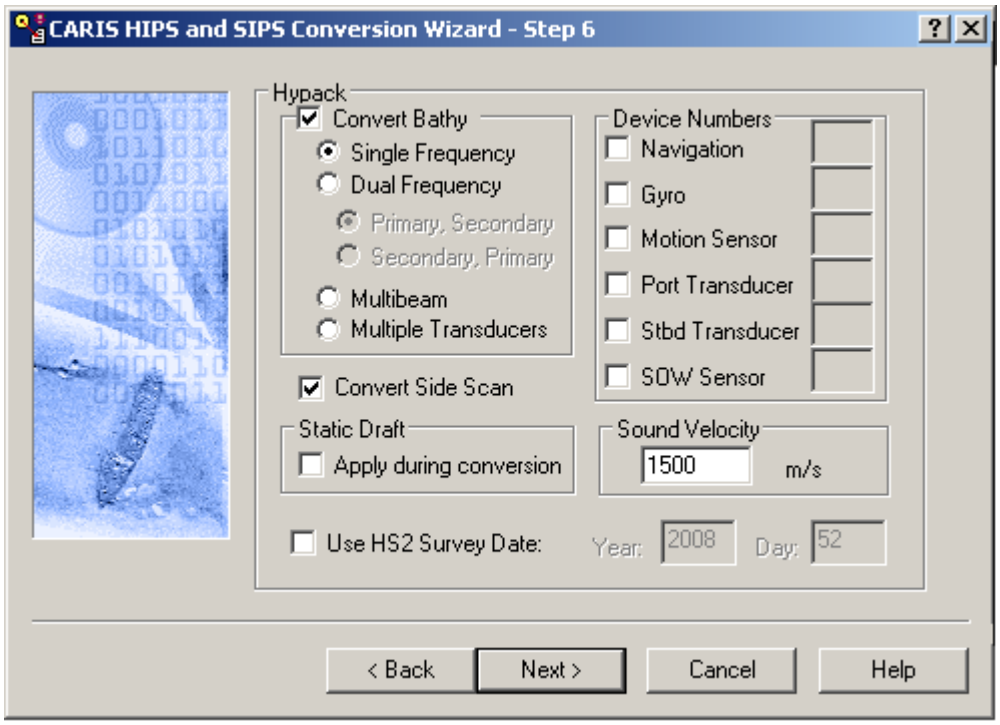
Horizontal and depth uncertainty values in the GSF file are also included during conversion.

Hypack

The Hypack format supports recording bathymetry from single beam, multibeam, and multiple transducer systems. It also supports reading side scan data from Hypack Max format.

For dual-frequency single beam data, you must indicate if the data records in Hypack are recorded as primary/secondary or secondary/primary. The primary frequency, by default, becomes the selected data for use in the field sheet.

If the conversion wizard detects records containing 12-bit data, they will be converted to 16-bit format. Otherwise they are converted to 8-bit data.



1. Select the type of data to be converted: Bathy, Side Scan or both.
2. Select a sounding data type:
   - single frequency
   - dual frequency (primary, secondary or secondary, primary)
   - multibeam
   - multiple transducers
3. If you wish to a apply the static draft during conversion, click the *Apply During Conversion* check box.

The *Device Numbers* section is used if sensor data was recorded from more than one device. To import data from a specific sensor,

you must type the number for that device. If no number is entered then *all* sensor data is imported. The following sensors are effected by the device number setting:

- *Navigation* for positional records.
- *Gyro* for heading records.
- *Motion Sensor* for roll, pitch and/or heave settings.
- *Port transducer* for port multibeam records.
- *Starboard transducer* for starboard multibeam records.
- *SOW Sensor* for speed over water records.

4. If data is recorded from more than one sensor device, click the sensor check box and enter a device number.
5. Enter the sound velocity value that is used to compute the original travel time data from the recorded depths. This allows sound velocity corrections to be applied in HIPS.

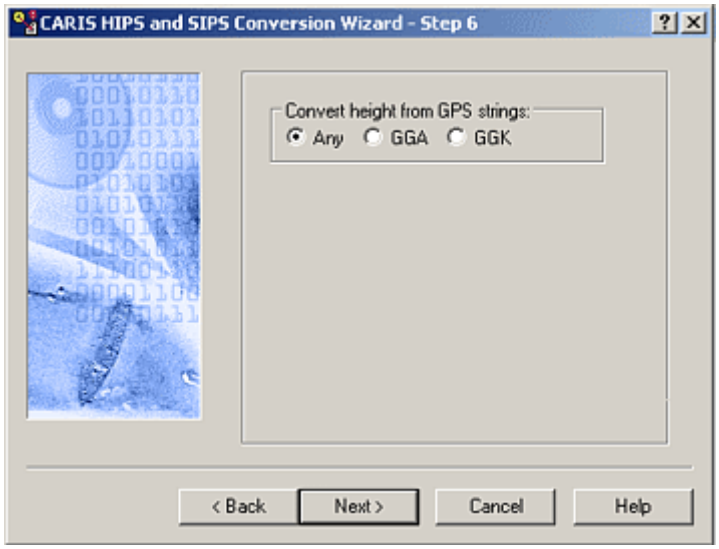
The HS2 format does not store day and year values with the timestamp (it does store a time value). When the data is converted to HIPS, the Day folder is used for reading the date. If you want to include another date value, you must complete the following *Year* and *Day* fields.

6. Select the *Use HS2 Survey Date* check box to enable the *Day* and *Year* fields.
7. Type the year and day in the fields.

Imagenex

To read GPS height data from Imagenex files:

1. Select the height string to be read: *GGA, GGK* or *Any*.



LADS

LADS data is contained in two files that are copied into the HIPS survey line folder: An ASCII CAF file that contains the bathymetry data, and a binary CBF file that contains the waveform data. The CAF file can contain data from more than one survey line (or "run"). When this is the case, the converter splits the file into separate line folders for each run. The .caf and .cbf prefixes are retained in the HIPS Line folder.

There are four sounding classifications in the LADS format:

- **S (Secondary sounding)**: the best soundings
- **P (Primary sounding)**: an unreliable result
- **N (No bottom found at depth)**: potentially a useful depth result
- **X (No bottom detected)**: no result

When the data is converted, each sounding is given a status flag according to its classification. The following table outlines classification and matching HIPS status flag.

LADS Classification	HIPS Sounding Flag	Definition
S	Accepted	Good soundings until they are flagged as rejected during cleaning in HIPS.
P	Rejected by disabled beam	A reject flag in HIPS that can be changed to Accepted, if necessary.
N	Examined	A special flag for an Accepted sounding. The sounding can be filtered out when creating a BASE Surface or exporting. It can also be flagged as Rejected during cleaning in HIPS
X	Rejected by Depth Gate	A type of Reject flag that can be changed to Accepted, if desired.

There are no dialog box options for the LADS format.

LAS

There are no options for the LAS format.

MarineSonics

There are no options for the MarineSonics format.

Navitronics

1. Select one of the following data types to be converted:
   - *Multibeam*
   - *Singlebeam*
   - *Multiple Transducer*
2. Select the *Convert Side Scan* check box to convert side scan data.

Navitronic side scan data can be imported in 16-bit format or converted to an 8-bit format.

3. Select either the *Preserve 16-bit* option or *Convert to 8-bit* option.

If the *Convert to 8-bit option* is selected, the *Scale* and *Shift* options are enabled. The *Scale* option averages the data and the *Shift* option selects a initial bit value (between 0 and 8) and includes the next seven bit values for export. For example, if you typed 8 as your initial value, then bits 8 to 15 are included in the export process.

4. Select the *Scale* option if you want to average the data into 8-bit format, or select *Shift* and type the value for the first bit.
5. Click **Next**.

Qmips

1. Choose a pair of sonar channels by clicking one of two options (1,2 or 3,4).

Reson PDS

There are no options for the Reson PDS format.

Scripps

There are no options for the Scripps format.

SDF

1. Select a Klien 3000 *High* or *Low* frequency.
2. Select a pressure sensor reading to set the depth of the towfish.
3. Hidden data consists of repeated pings over the same area. Select the *Convert Hidden* check box to include hidden data in conversion.
4. Select the *Convert SSGyro* check box to include side scan sonar gyro data.
5. Select the *Cable Out is Horizontal Layback* option to omit sensor depth during conversion. If there is no sensor depth and the towpoint has a Z-offset of zero, the cable out value is treated as the horizontal layback value during processing.

SDF side scan data can be in 16-bit or 8-bit format. If the data is in 16-bit format, you can choose to convert to an 8-bit format.

6. Select either the *Preserve 16-bit option* or *Convert to 8-bit* option.

If the *Convert to 8-bit option* is selected, the *Scale* and *Shift* options are enabled. The *Scale* option averages the data and the *Shift* option selects a initial bit value (between 0 and 8) and includes the next seven bit values for export. For example, if you typed 8 as your initial value, then bits 8 to 15 are included in the export process.

7. Select the *Scale* option if you want to average the data into 8-bit format, or select *Shift* and type the value for the first bit.

Seabeam

1. Make sure the *Convert Side Scan* option is checked to include side scan data with bathymetric data.
2. Select the *Shorten Line Names* check box to modify the file name to a 12-character CARIS name (YYDDD_HHMMDD).

Seafalcon

Seafalcon allows the option of importing a full-sounding dataset or a thinned dataset. The thinning options are based on reducing the number of beams per profile that are imported. All profiles are imported.

1. Select the thinning option, if needed.

The following two options are enabled if the thinning option is selected.

2. Select the amount of data to be processed.
   - half the number of beams are imported
   - one fourth the number of beams from each profile are imported
3. Select the sounding type to be imported into HIPS format.
   - shoal
   - average
   - deep

You can select the swaths per ping that you want to import. In deep water, the Seafalcon sonar produces five swaths with each firing of the transducer. These five swaths are in an along-track position to the ship—with swath 1 forward of the sonar and swath 5 aft of the vessel. In shallow water only one swath per ping is generated.

By default all five swaths are selected for import, but you can decide which swaths to import.

4. Select the swath number(s) for import.

SEGY

There are no options for the SEGY format.

SHOALS

All soundings in the SHOALS format are given a confidence value during acquisition. These confidence values are used to flag soundings during conversion so that soundings with certain confidence values are rejected. The following status flags are assigned to soundings. These can be changed in HIPS.

- **Rejected by disabled beam**: This flag rejects soundings with a negative confidence value.
- **Rejected by depth gate**: This flag is for soundings with confidence values that range from 0 to 70. The 0 to 50 range represents soundings where no bottom was found, and the 51 to 70 represents soundings of questionable confidence.
- **Accepted**: This flag is for soundings with confidence values from 71 to 99.

When converting to HIPS, all soundings with a confidence value less than the *Confidence Cutoff Value* will be automatically rejected. The default cutoff value is 70.

1. Use the mouse to select a cutoff value.

If the dataset contains land data, you can select the topographic format (TOF) options to convert this data too.

2. Select either the *First Pulse* check box to convert the return from the top of the vegetation canopy or the *Second Pulse* check box to convert the return from the ground.

Simrad

1. If you are using the EM3000 sonar, select the channel used for the primary position.
2. Select *Manual Override* to read data from the navigation system that is marked inactive by Simrad.
3. Click the *Side Scan/Backscatter* box if you want to convert this data.

You can fit critical line information into the 12 characters provided by the CARIS Source ID attribute for soundings.

4. Select the *Shorten Line Names* check box to modify the file name to a 12-character CARIS name (YYDDD_HHMMDD).

Large attitude data files can result if there is a straight one-to-one import into HIPS. The *Attitude Data Decimation* option down-samples the data so that a smaller attitude dataset is converted. The *Decimation Factor* determines the ratio of attitude data. For example, if you select 10 as the *Decimation Factor*, then one out every tenth attitude record is converted.

5. Select the *Attitude Data Decimation* check box to implement this option.
6. Select a *Decimation Factor* to set the ratio for down-sampling attitude data.

Motion data (heading, heave, roll, and pitch) is automatically read from the active motion reference unit. However, you can select another motion reference unit to read the data. Gyro data can also be read from the headings datagram.

7. For *Heading*, select the *Auto* option to read gyro data from the active motion reference unit, or select an inactive unit by selecting *MRU1*/*MRU2* options. Select the *H. Diagrams* option to read gyro from the headings datagram.
8. For *Heave*, *Roll*, and *Pitch*, select the *Auto* option to read data from the active motion reference unit, or select an inactive unit by selecting *AutoMRU1*/*MRU2* options.
9. For *GPS Height*, select a source from either the *EM Height* datagrams or the *GPS String* in the navigation datagrams.

Navigation time stamps can be selected from either the logging system or from a source specified by Simrad Installation datagram (the GPS string in the navigation datagram or the logging system).

10. Choose *System* to select the logging system or choose *Automatic* to select the timestamp specified by the Installation datagram.
11. If you are converting single beam data from the dual frequency EA400\600 system, select *Primary* to use the primary sounding as the selected sounding in the HIPS file, or select *Secondary* to use the secondary sounding as the selected sounding.

SWATHPlus

1. Type a value in the *Speed of sound* field to apply sound velocity to the imported data.

UNB

1. Select one or two transducers.
2. Select the *Convert Sonar* option, if you want to import side scan data.

Winfrog

1. Select *Single* or *Dual Frequency*.
2. Select attitude records to be converted.
3. Specify the speed of sound, so that slant range data can be generated from the measured sounding depths.
4. Choose the navigation records to be converted.

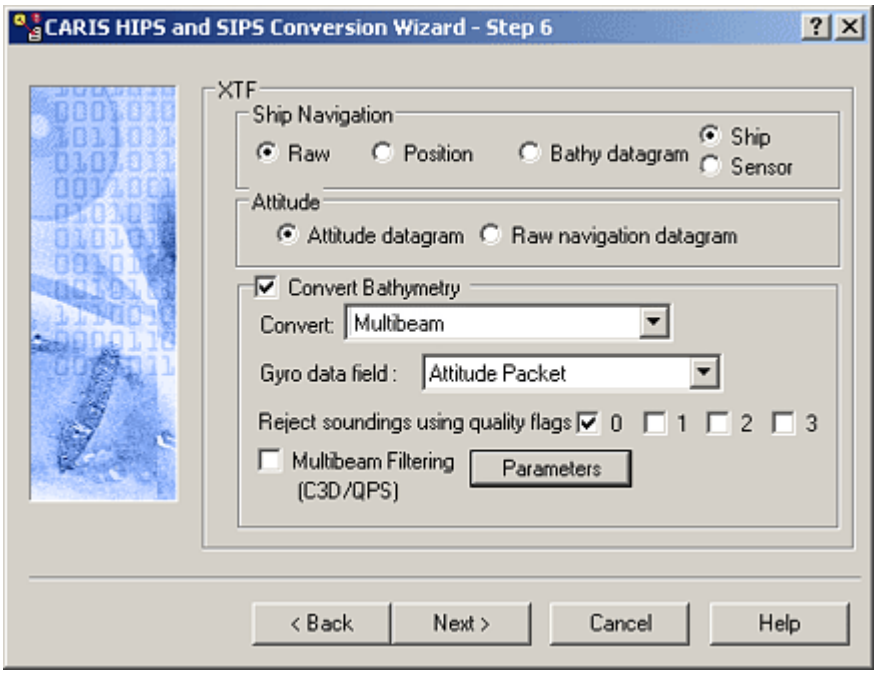
XTF

The XTF format conversion has two sections: bathymetry and side scan.

.

For both sonar types, you must indicate the field used as the source for navigation and gyro/heading data.

For side scan, there are additional options for converting horizontal tow fish layback or tow cable length distances. There are also options for reading the dynamic depth information for a remote operated vehicle (ROV).



1. Click one of following options to select a source for reading the navigation data fields:
   - *Raw* navigation datagram
   - *Position*
   - *Bathy datagram*

If you select *Bathy datagram*, the *Ship* and *Sensor* fields are enabled.

2. Select either the *ship* or *sensor* as a source for reading navigation data.

Note: Navigation data for the ship can now be read from the new XTFRAWNAVIGATION datagram that is available for Reson MBES data stored in XTF format by Triton ISIS.

If you select the *Raw* navigation option, the option to read the ship attitude data from the historical XTF attitude datagram or the new XTFRAWNAVIGATION datagram is activated.

3. Select one of the following options:
   - attitude datagram
   - raw navigation datagram

Convert Bathymetric data

To convert bathymetry data

4. Click the *Convert Bathymetry* check box.
5. Select a bathymetric data format from the *Convert* drop-down list:
   - Multibeam
   - Multi-transducer
   - Single beam from records *{Prim=0}* or *{Prim=1}*:

     If there are two channels of data, you can identify which channel contains the primary data
   - Single beam from AUX (1,2,3, or 4).

Note: Benthos C3D data will be converted to two lines: multibeam and single beam.

The singlebeam HIPS line (with the addition of _SingleBeam to the line name) will have the same set of navigation and attitude data as the multibeam line, but the sounding depths will be single beam data from the XTF sensor AuxAltitude field.

Select gyro data field

The *Gyro data field* source fields are enabled. One source is *CMG (Course Made Good) from Navigation*. Other options in the pull-down list are determined by the selections made in the previous steps.

If you previously selected the *Raw navigation datagram* option to read navigation and attitude data then the option to use the same datagram for ship gyro data is available on the drop-down list.

If you previously selected the *Attitude datagram* option then the option to use that datagram is available.

If you selected either the *Position* or *Bathy datagram* options at Step 1, you can select from the following options:

- ship
- sensor

- attitude packet
- Course Made Good (CMG) from Navigation
- Course Made Good from SSS Navigation

Set reject soundings flags

The XTF format attaches quality flags to soundings. For example, with Reson sonars, 0 represents the lowest quality and 3 represents the highest quality. You can reject soundings with a specific flag. These soundings are flagged as Rejected by Disabled Beam in HIPS.

6. Select a quality flag so the soundings with that specific flag value are rejected when converted to HIPS format.

If you are converting C3D or QPS data, you can apply filtering before moving to the next step.

If you do not want to apply filtering

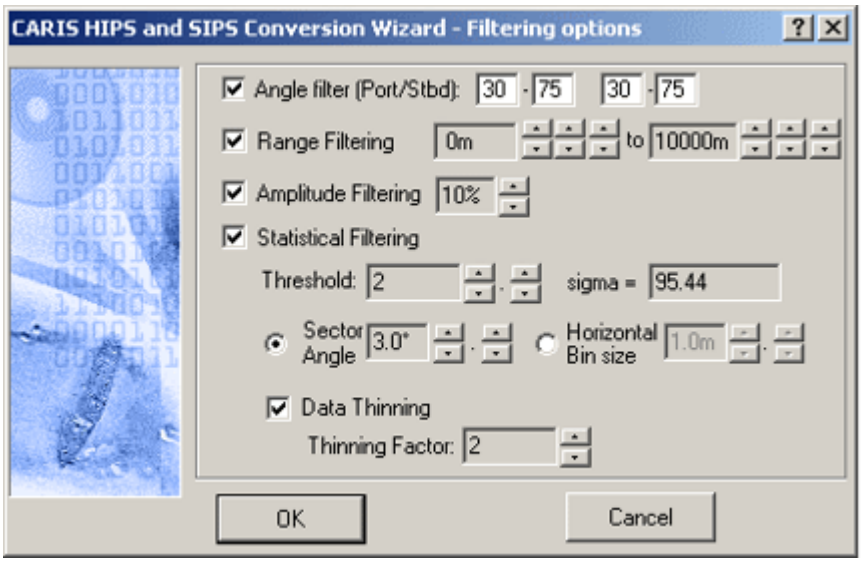
7. Click **Next** to continue with the import process.

Filtering

To apply filters for Benthos C3D or QINSy QPS data

8. Select the *Multibeam Filtering* check box and click **Parameters** to select filtering options.

The Filtering options dialog box is displayed.



Benthos C3D data can be filtered so that only the best quality data is imported into HIPS. You can apply the filter to port/starboard beam angles.

1. Type a range of beam angles to select the cleanest data from the port and starboard beams.

All other options are applied to QPS data only.

The *Range Filtering* option converts soundings within a specified distance (in metres). Soundings outside of this range are not converted.

2. Select the *Range Filtering* check box to implement this option.
3. Click the up or down arrow buttons to select a minimum and maximum distance.

The *Amplitude Filtering* option filters soundings according to amplitude value. For each ping (port and starboard pings are handled separately), the min/max amplitude values are obtained, and samples that fall below the selected percentage (0-50%) are rejected.

4. Select the Amplitude Filtering check box to implement this option.
5. Click the up or down arrow buttons to select a percentage value for rejecting soundings.

The *Statistical Filtering* option controls which soundings in each swath are considered for conversion. This option calculates the mean depth and standard deviation within a swath sector or horizontal bin. It then prevents any soundings that fall outside a multiple of the standard deviation from being converted.

If Statistical Filtering is used, then the mean within the sector is re-computed. Ultimately, the soundings are sorted by their residual from the mean and those closest to the mean are converted. The actual number of soundings converted from within each sector is determined by the thinning factor (if used).

6. Select the *Statistical Filtering* check box to implement this option.
7. Determine a *Threshold* (multiples of the standard deviation) value by clicking the up and down arrow buttons.

The equivalent confidence value is displayed as a percentage.

The filtering and thinning methods are executed within a sector angle interval or a horizontal bin size. The sector angle option divides the swath into sectors according to degree angles while the horizontal bin size divides the swath into horizontal sectors based on a specified across-track distance.

Vessel motion and transducer mounting angles are considered when sector angle and horizontal bin locations are determined.

1. Select the *Sector Angle Interval* option and choose a degree level (to a maximum of 10º) by clicking the up or down arrow buttons.
2. As an alternate method, Select the *Horizontal Bin Size* option choose an across-track distance by clicking the up or down arrow buttons.

The thinning factor reduces the number of soundings converted on a per swath basis. The thinning factor uses a power of two reduction control (1/2, 1/4, 1/8, etc.) so that one of two, one of four or one of eight soundings can optionally be converted per swath sector.

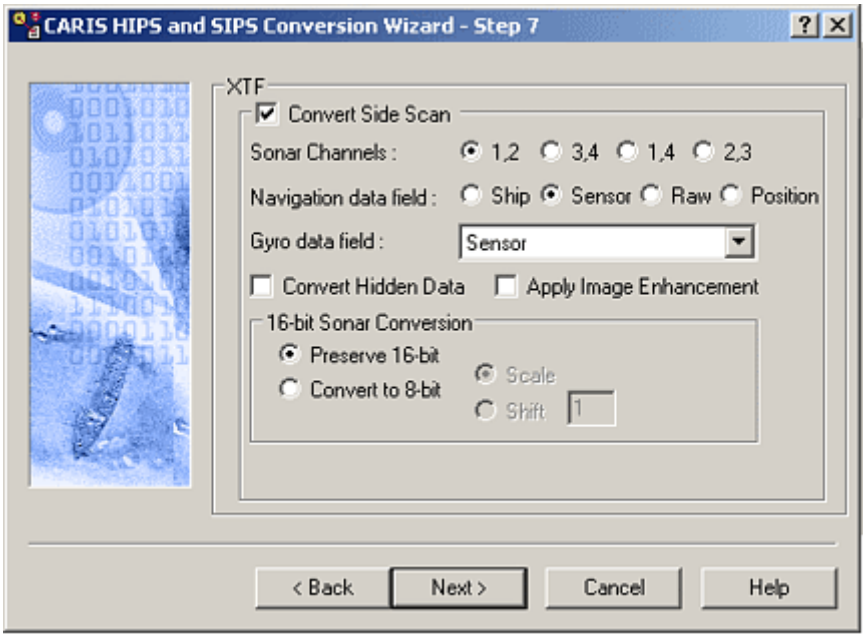
3. Select the *Data Thinning* check box to implement the thinning option.
4. Select a *Thinning Factor* value by clicking the up or down arrow buttons.

5. Click **Next** to go to the Convert Side Scan Data dialog box.

Convert Side Scan data

The XTF dialog box for converting side scan is displayed.

If you do not want to convert side scan data then click **Next** and go to "CONVERT DATA" ON PAGE 177.



Note: Contact records created in Triton ISIS are also imported into HIPS when associated XTF data is imported.

To convert side scan data,

6. Click the *Convert Side Scan* check box.
7. Select one of four sonar channels by clicking an option.
8. Select one of following options to select a source for reading the navigation data for the towfish:
   - ship
   - sensor
   - raw
   - position
9. Select one of the following options from the pull-down list as a source for reading the gyro fields for the towfish:
   - ships
   - sensor
   - attitude packet

- raw navigation
- Course Made Good from Navigation
- Course Made Good from SSS Navigation

The XTF format flags repeated pings over an area as hidden data.

10. Select the *Convert Hidden Data* check box to include this data type in the SIPS project.
11. To enhance the image by scaling the intensity values, click the *Apply Image Enhancement* check box.

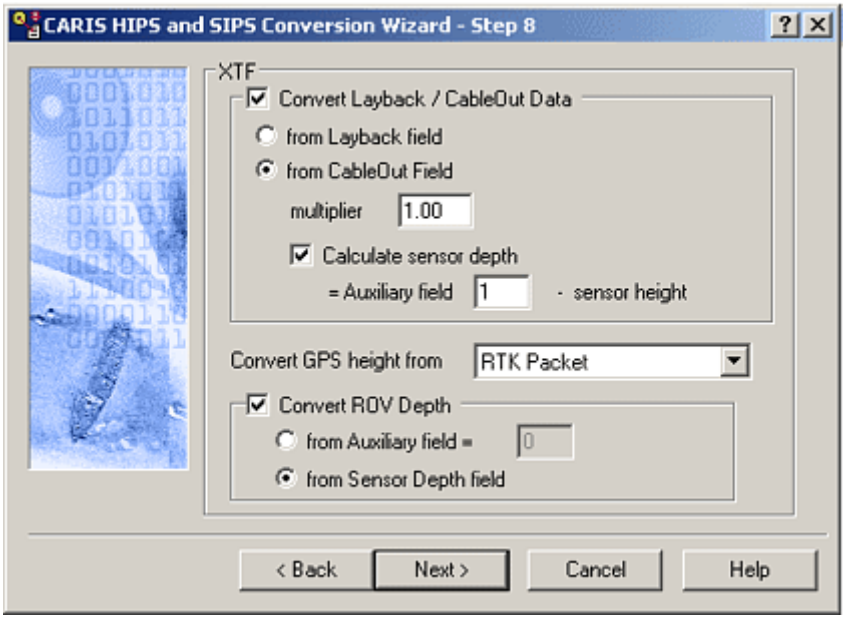
XTF side scan data can be in 16-bit or 8-bit format. If the data is in 16-bit format, you can choose to convert it to 8-bit format.

12. Select either the *Preserve 16-bit option* or *Convert to 8-bit* option.

If the *Convert to 8-bit option* is selected, the *Scale* and *Shift* options are enabled. The *Scale* option averages the data and the *Shift* option selects a initial bit value (between 0 and 8) and includes the next seven bit values for export. For example, if you typed 8 as your initial value, then bits 8 to 15 are included in the export process.

13. Select the *Scale* option if you want to average the data into 8-bit format, or select *Shift* and type the value for the first bit.
14. Click **Next**.

The third XTF conversion dialog box is displayed.



15. If you want to recompute side scan towfish positions in HIPS with the recorded horizontal layback or tow cable length data, make sure the *Cable Out/Layback* box is checked.
16. Select one of the following options:
    - *from Layback field*: The distance from the vessel towpoint to the towfish.

- *from Cable Out field*: The length of the tow cable to the towfish

17. If you selected *Cable Out*, enter a number in the *Multiplier* box to correct for integer value recording, if necessary.
18. Select the *Calculate Sensor Depth* check box to enable options for calculating the depth of the towfish below the waterline.
19. Type a number of the *Auxiliary* field that contains the total measured depth.

The total measured depth and the digitized height of the towfish are used to compute the depth of the towfish below the waterline. The towfish depth can then be combined with the tow cable length data and the vessel's tow point configuration to compute the position of the towfish.

When ship navigation is being read from the bathymetry packet, then the options available for the source of GPS ellipsoid height data are the *RTK Packet* and the *Ocean Tide* field in XTF.

However, if the ship navigation is being read from the new XTFRAWNAVIGATION datagram then the GPS ellipsoid height, if available, will always come from this new raw navigation datagram.

20. Convert the GPS height of the antenna or navigation sensor from:
    - RTK packet
    - Ocean Tide
21. If data is being collected from an ROV (Remote Operated Vehicle), click the check box to enable the source options for measurement of vehicle depth.

The auxiliary and sensor depth fields are enabled.

22. Click the appropriate check box to select a source for reading ROV depth data:
    - Auxiliary
    - Sensor Depth
23. Type the number of the *Auxiliary* field (from 1 to 6), if this option is selected.
24. Click **Next** to go to the step to "CONVERT DATA" ON PAGE 177 of the Conversion Wizard.

8

Survey Line Processing

Select Survey Lines

See "SELECTION TOOLS" ON PAGE 42 for more information on selecting track lines.

You must select a survey line to use these editor tools:

- Attitude Editor
- Navigation Editor
- Single Beam Editor
- Swath Editor
- Side Scan Editor

You can select one or more survey lines to use these tools and processes:

- single beam filters
- swath filters
- Total Propagated Error (TPE) filters
- restart cleaning
- load tide
- sound velocity corrections
- compute GPS tide
- compute TPE
- load delta draft
- load TrueHeave
- close track lines
- merge
- recompute towfish navigation
- slant range correction

You can let the program automatically select survey lines for these functions:

- BASE Surface
- mosaicking

Query Line Data

Examine information about one or more selected track lines.

1. Select track lines in the Display window or in the Project tab of the Control window.
2. Select the command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

The Query view in the Worksheet window can display more than 40 different fields of data about each line. You can view all query fields by selecting *Detailed Line Query* in the General Options dialog box (see "GENERAL" ON PAGE 75) or just the most important fields by de-selecting this option. The information is read-only and cannot be modified.(see "WORKSHEET WINDOW" ON PAGE 15).

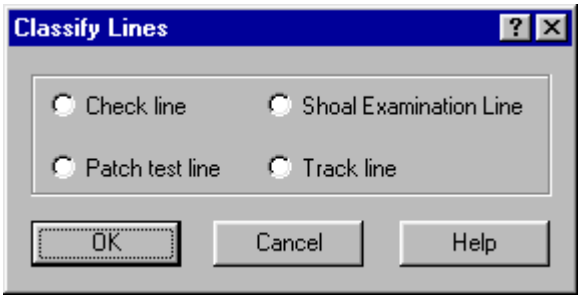
Set Line Classification

Set or change a classification flag for a track line.

1. Make sure the Ship Track Lines layer is selected in the Control window.
2. Select a track line in the Display window or Control window.
3. Select the command.

Menu: Edit > Classify Lines

The Classify Lines dialog box is displayed.



4. Select from four options for setting the line classification flag:
   - *Check Line*: A line that is run perpendicular to the survey and is used to check the validity of survey data.
   - *Patch Test Line*: A line that is run over a surveyed area to calibrate sensor and sonar offsets.
   - *Shoal Examination Line*: A line that is run over a surveyed area to re-check possible shoals on the sea floor.
   - *Track Line*: A line that is part of the main survey grid.
5. Click **OK**.

The line is now flagged according to one of four categories.

View Rejected Soundings

You can show or hide rejected data in the Attitude, Navigation, Single Beam, Swath, and Subset Editors. Data that was rejected in the Subset Editor can be viewed in the Swath Editor.

1. Select the command.

Menu	View > Show Rejected
Tool	
Key	<F4>

You can also control which type of rejected data is displayed in the Single Beam, Swath, and Subset Editors.

2. Select the View command.

Menu	View > Rejected Flag Types

The Rejected Flag Types dialog box opens.

3. Place a check mark beside the box with the type of rejected data you want to view.
   - *Rejected by Swath/Single Beam Editor*: View sounding data rejected in the Swath or Single Beam Editors or by the Swath or Single Beam filters.
   - *Rejected by Hydrographer*: View data rejected interactively in the Subset Editor by the hydrographer.
   - *Rejected by TPE*: View data rejected through the Total Propagated Error filtering process.
   - *Rejected by Surface Cleaning*: View data rejected through the Surface Cleaning process.
   - *Rejected by Surface Filter*: View data rejected through surface filtering.
   - *Rejected by Depth Filter*: View data rejected by the depth filter in the Conversion Wizard or the Generic Data Parser when it was converted to HIPS format.
   - *Rejected by Disabled Beam*: View data rejected due to a disabled beam status during conversion. This occurs when the beam is disabled in the vessel configuration or if the sonar system flagged the beam as a bad detection in the raw data file.
4. Click **OK**.

The rejected data types you selected are displayed in the editors.

Designated Soundings

The Designated Sounding flag identifies the shallowest sounding on important features. For example, in a cluster of soundings surrounding an outcrop of rocks, the shallowest sounding among that cluster would be identified as Designated.

The purpose of the Designated Sounding flag is to ensure that the shallowest depths over significant seabed features are maintained for standard hydrographic products.

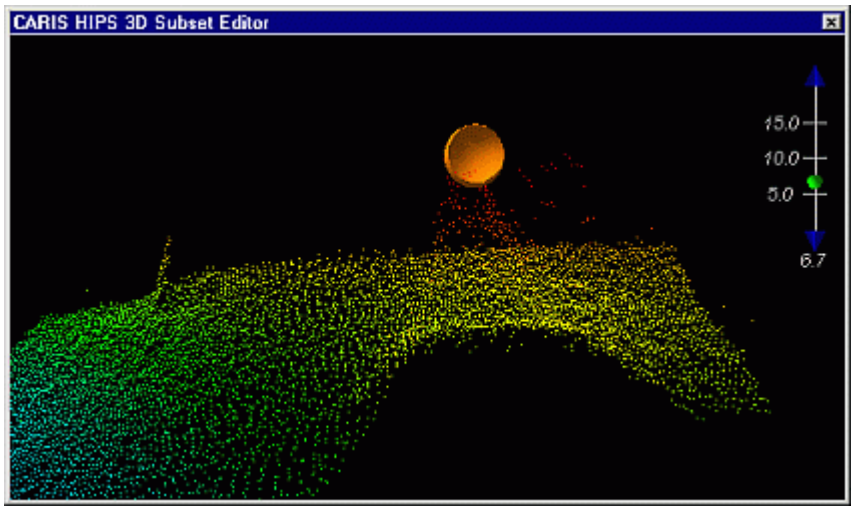
When these soundings are applied to a Bathymetry Associated with Statistical Error (BASE) Surface, the Surface is regenerated so that the Designated Sounding's depth value is applied to the nearest node (see "FINALIZE BASE SURFACE" ON PAGE 427).

The Designated Sounding option is available in the Swath and Subset Editors. Designated Soundings can exported to ASCII, CARIS, GSF, and HOB formats. The location of designated soundings can be viewed in the Display window (see "CRITICAL SOUNDINGS" ON PAGE 210).

1. Select the shallowest sounding from a cluster of soundings around a feature in the Swath or Subset Editors.
2. Select the Designated Soundings command.

Menu	Edit > Status Flag > Designated
Tool	
Key	<D>

When a sounding is flagged as Designated, it is displayed in the Subset or Swath Editor by a special symbol as shown below.



Find Designated Sounding

The Find Designated Sounding command automatically selects the shallowest sounding in cluster of highlighted soundings. This feature reduces the time needed to select a Designated Sounding.

1. Use the cursor to highlight soundings in the Swath, Single Beam or Subset Editors.

Menu	Edit > Status Flag > Find Designated
Tool	

2. Select the Find Designated Soundings command.

The shallowest sounding among the cluster of highlighted soundings is flagged as Designated.

Critical Soundings

Display soundings with Designated, Examined and Outstanding status flags.

For more information on sounding status flags, see "HIPS STATUS FLAGS" ON PAGE 623.

The Display Critical Soundings command reduces the time needed to locate Designated, Examined and Outstanding soundings when working with large datasets. These soundings are displayed using a HCSIDX file that contains the line, profile, beam number and depth of all special soundings in the project.

The HCSIDX file is a project-level file that includes all special soundings. You do not need to select a track line to generate or update the file.

1. Select the Display Critical Soundings command.

Menu	View > Display Critical Soundings

If you are selecting this command for first time, a message is displayed informing you that the file has not yet been built and that building this file might take time. If a file already exists, then it is automatically compiled.

When the file is built or rebuilt a critical soundings layer is created and displayed in the Session tab of the Control window.

2. Select the Critical Soundings layer and select the Refresh command.

The soundings are drawn in the Display window according to the colour settings for each sounding flag (see "STATUS" ON PAGE 92).

3. To view information on specific soundings, select the soundings in the Display window and select a Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

Information about the soundings is displayed in the Query tab of the Worksheet window.

Critical Sounding Properties

Critical soundings can be colour-coded according to their specific status flags using the Options dialog box (see "STATUS" ON PAGE 92) or the Properties window. You can also change the font and type size for soundings from the Properties window.

1. To colour code the soundings, select the Display tab in the Options dialog box and select Status from the list, or select the Properties window.
2. To change the type face and font size of the soundings, click the Font tab in the Properties window to view the standard Windows Font dialog box.

Menu	Window > Properties
Tool	

Reject Line

Use the Reject Line command to remove an entire track line from data processing.

When a rejected line is merged, the profiles are marked as rejected, which prevents the soundings from being used in BASE surfacing, surface cleaning, and mosaicking. However, status flags on the individual soundings are maintained. A rejected line is not processed in the tiling operations in the CARIS Field Sheet Editor or exported to a CARIS map or ASCII file.

1. Select a track line.

Menu | Edit > Reject Line

2. Select the Reject Line command.

The rejected line is displayed in the Project tab in the Control window with a red X beside the file name.

3. To change a rejected line back to accepted, select the Accept Line command.

Menu | Edit > Accept Line

The line is displayed in the Project tab of the Control window with a red exclamation point (!) beside the file name. This indicates the line has been flagged as outdated. You must merge the soundings again before they are available for further processing (see "MERGE" ON PAGE 253).

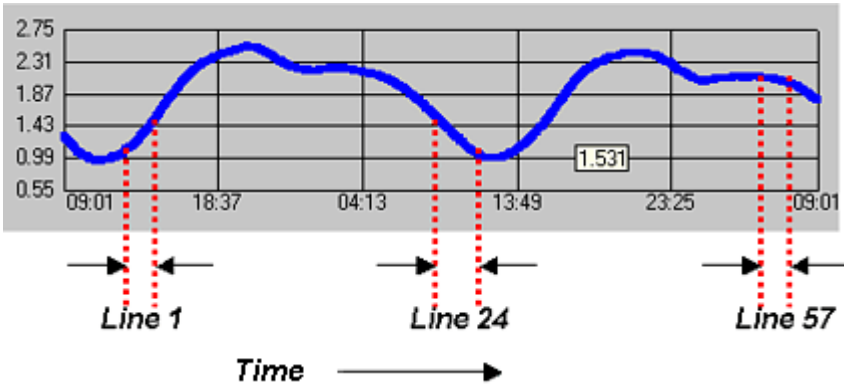
Correct for Tide

Tidal observation data must be loaded for every track line before the Merge process can be executed. Tide data is used to generate final depths relative to the tide datum by subtracting the tide from the sounding depth.

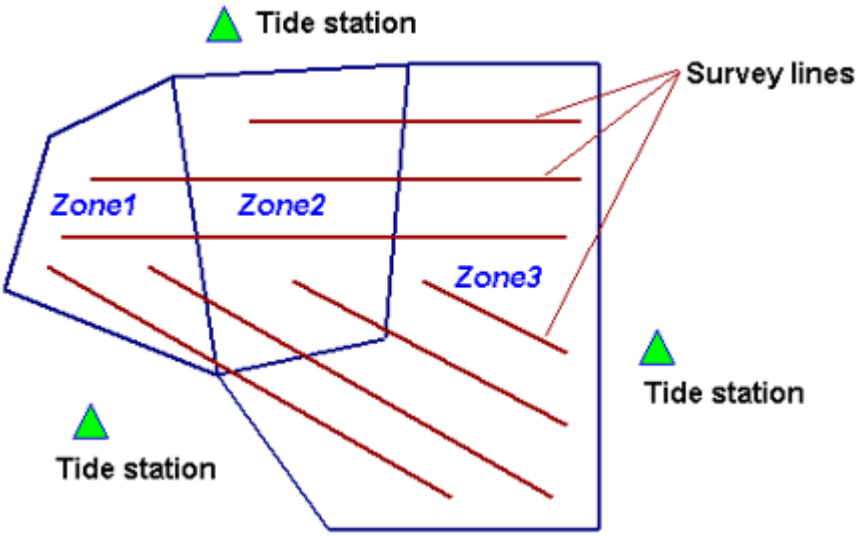
There are two options for tide loading:

- Apply tide observations from a *single* station to a track line or lines using the time extent of the line(s).

The illustration below shows the tide values from a single station that would be applied to three selected lines, based on the time each line was recorded.



- Apply tide observations from *multiple* tide stations to a line or lines, using tide zones. Where a survey line falls within two or more tide zones, the tide data to be loaded for each part of a line is determined by the zone it falls in (as illustrated below).



The tide data file must span the entire time frame of the track line. The time zone of the tidal observation data must match the time zone of the survey data to which it will be applied.

The output from the Load Tide function (see "LOAD TIDE" ON PAGE 215) attaches a binary HIPS tide-time series to each track line

Tide Files

Tide data from a single station is loaded from files with the extension *.tid. These files can be created or edited in a text editor (such as Notepad), or with the Tide Editor.

HIPS supports a standard Canadian Hydrographic Service tide format called COWLIS as well as two NOS/NOAA formats. HIPS also supports a basic format consisting of just the required date, time, and tide values.

Track lines in areas without water level changes, such as rivers or lakes, must also have a tide file. In this case, or if tidal data cannot be obtained, a "zero tide" file—where all tide/time values are zero—is used so merging can proceed.

All tide files, including zerotide.tid, are located by default in…\Hips*ver*\Tide.

For more detail, see "TIDE FILE FORMATS" ON PAGE 647.

Tide Zone Files

The tide zone option assigns tide observations to track lines from multiple tide stations based on the time period in defined geographic zones.

Each tide zone is defined by a closed polygon with tide, time and range corrections for a primary station, plus up to three backup secondary tide stations.

HIPS uses a Zone Definition File (with the extension .zdf) for defining tide zone data. This is an ASCII file and can be prepared with any text editor.

For complete description of the ZDF format, see "TIDE ZONE DEFINITION FILES" ON PAGE 653.

Display tide zones

Tide zones and station locations can be displayed. To view an outline of tide zones:

Menu	View > Display Tide Zones

1. Select the Display Tide Zones command.

The Open Tide Zone Definition File dialog box is displayed.

2. Select a tide zone file, or type the file name and path in the *File Name* field.
3. Click **Open**.

The outlines of the tide zones, and their labels, are drawn in the Display window. The Tide Zones layer is listed in the Session tab of the Control window.

Tide stations and weighted average

When an average tide is calculated from multiple tide stations, the weight given to a tide station's data is inversely proportional to its distance from the vessel at that time. In other words—the further a station is from the survey line, the less weight is accorded to the data from that station.

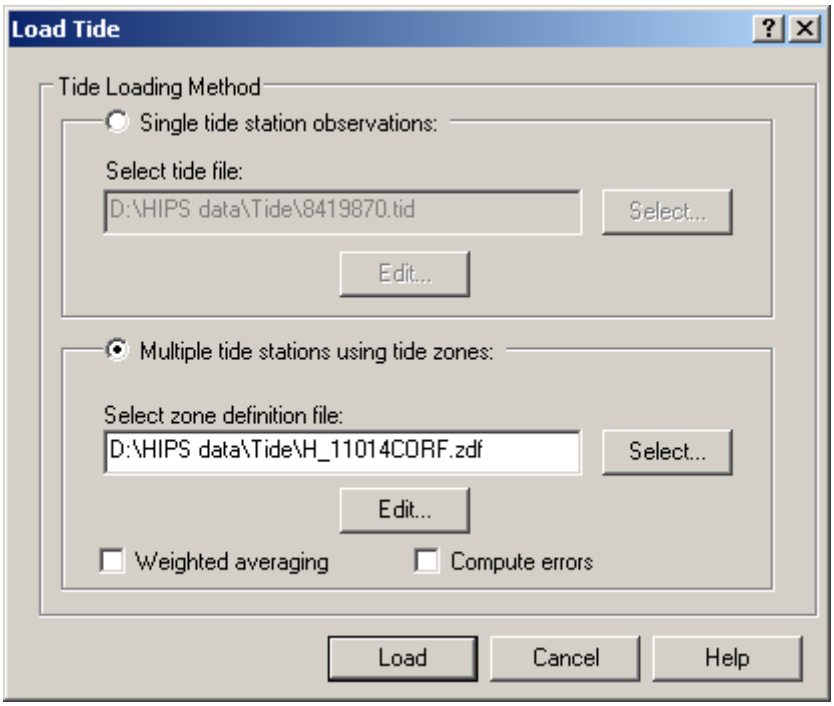
Load Tide

Apply tidal observation data to selected HIPS track lines. A tide zoning option is available with or without weighted averaging from multiple tide stations.

1. Select the track line(s).
2. Select the Load Tide command.

Menu	Process > Load Tide
Tool	

The Load Tide dialog box is displayed.



Select tide files

Choose a tide file containing tidal observations from a single tide station for loading into the currently selected lines.

1. Choose the *Single Tide Station Observations* option.
2. Click **Select** to choose a tide (.tid) file.

The Open Tide File dialog box is displayed.

3. Select a tide file or type the file path and name in the field.
4. Click **Open**.
5. The tide file name is displayed in the Select Tide File field.
6. To revise the tide file, click **Edit** to open the Tide Editor.
7. Click **Load** to associate the file with the selected track line(s).

Select tide zone files

Choose a tide zone file to be opened into the currently selected lines.

1. Choose the *Multiple Tide Stations Using Tide Zones* option.

2. To choose a tide zone (.zdf) file, click **Select**.

The Load Zone Definition File dialog box is displayed.

3. Select a tide file or type the file path in the field.
4. Click **Open**.
5. The tide zone file is displayed in the Select Zone Definition File field.
6. To revise data in the zone definition file, click **Edit** to open a text editor.
7. Select the *Weighted Averaging* check box to apply this option (see "TIDE STATIONS AND WEIGHTED AVERAGE" ON PAGE 214).
8. Select the *Compute Errors* check box to apply the uncertainty value in the tide zone file. The uncertainty value is used to create a tide zone error.
9. Click **Load**.

The tidal observation is loaded into the selected track lines using the tide zone definition file.

Attitude Sensor Data

You can examine and clean the motion data of the vessel or towed transducer using the Attitude Editor. For more information on editing attitude data see "ATTITUDE EDITOR" ON PAGE 271.

Attitude Filter

Apply changes to attitude data across entire track line(s). The filter can be used to accomplish two tasks:

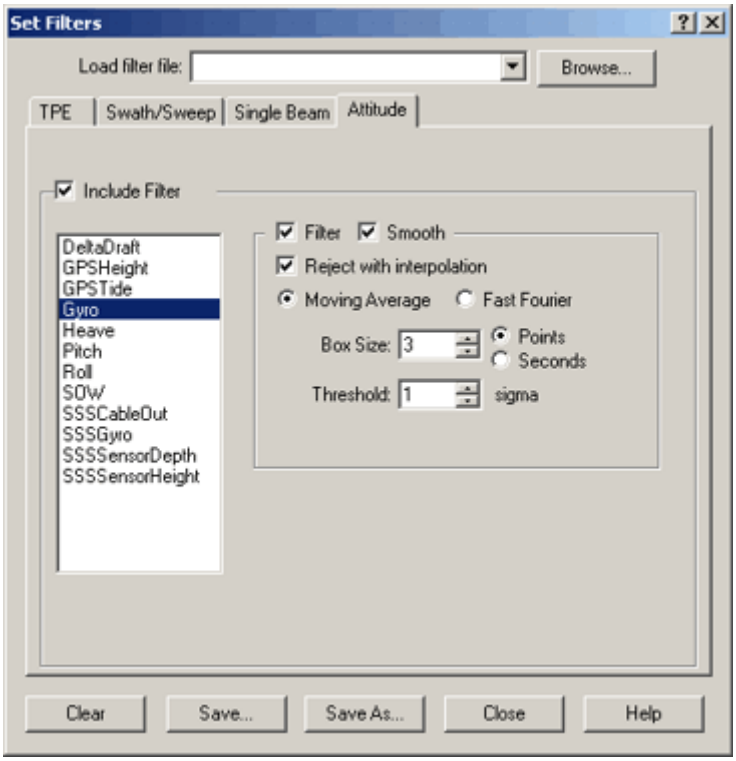
- **Smooth data**: The Smooth function is used to smooth out localized variability. The parameters for creating the smoothed data are saved to the SmoothedCoefficients file in the HCDS_Data\Project\Vessel\Day\Line folder. This file is applied to the track line during any process that supports smoothing.
- **Filter data**: The Filter function is used to reject attitude that falls outside of defined boundaries. Soundings with the same time stamp as the rejected data are also flagged as rejected. You can choose to reject attitude data with interpolation.

These options are also found in the Attitude Editor (see "ATTITUDE EDITOR" ON PAGE 271).

1. Select a track line.
2. Select the Set Filters command.

Menu	Tools > Set Filters
Tool	

The Set Filters dialog box is displayed.



1. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

If you chose to load a filter file, all the values used in that file are shown in the fields. You can filter data or change any of the values in the fields.

2. Select the *Include Filter* option to make the list of data types that can be processed by the Attitude filter.
3. Select the *Filter* or *Smooth* check boxes (or both) to activate the options. If you select the *Filter* check box, the *Reject with Interpolation* check box is also active.
4. Select *Reject with Interpolation* check box so that soundings with the same time stamp as the rejected attitude data are available for processing. If this check box is cleared, the soundings are flagged as rejected (break interpolation). This check box is active only if the *Filter* option is selected.
5. Filter/Smooth data by selecting either the *Moving Average* or *Fast Fourier* methods:
   - The *Moving Average* function calculates an average sensor value over a window of data using the *Box Size* parameter. The window is defined as a number of data points or seconds centred on a point. The average value for all of the values in the window is calculated and will be given to the central point.
   - The *Fast Fourier* transformation performs a low-pass filter on the selected sensor. You select a *Box Size* in either points or seconds.

This value is converted into seconds and then inverted to become the cut-off frequency.

6. Select the *Box Size* parameter by clicking the up or down arrow buttons. When using the *Moving Average*, the *Box Size* determines the size of the window for averaging the data point. When using the *Fast Fourier*, the *Box Size* determines the cut-off frequency that is applied to all data.
7. Determine if the *Box Size* is in (data) points or time, by selecting the *Points* or *Seconds* options.
8. Set the *Threshold* for smoothing or filtering data by clicking the up or down arrow buttons. The *Threshold* is a multiple of the standard deviation (sigma).
9. Click **Close**, or click **Clear** to remove data from the fields.
10. Click **Save As** to save the settings to a filter file, or click **Save** to settings to an existing file.

TrueHeave Data

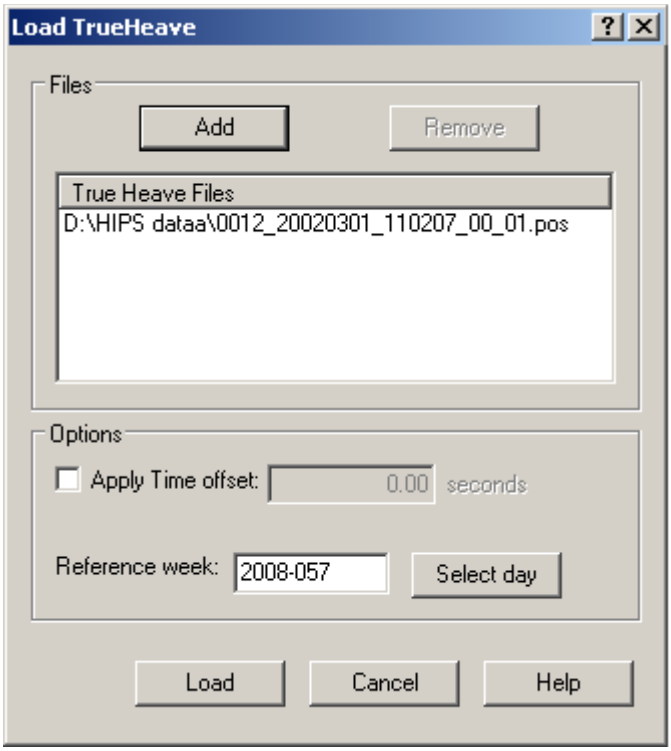
Load TrueHeave data to a survey track line.

Applanix's TrueHeave values are computed from real-time heave data and stored in a separate binary file along with the corresponding heave data. This file is not imported into HIPS during regular data conversion and has to be separately loaded to the track line. The location of TrueHeave files can be set from the Directories tab of the Options dialog box (see "DIRECTORIES" ON PAGE 97).

Menu	Process > Load TrueHeave
Tool	

1. Select a track line.
2. Select the Load TrueHeave command.

The Load TrueHeave dialog box is displayed.



3. Click **Add** and select the TrueHeave file.

The file name is displayed in the dialog box.

4. To remove a file, select it and click **Remove**.
5. Select the *Apply Time Offset* if the time stamps in the TrueHeave files do not match the time stamps for the rest of the project.
6. Type the offset to synchronize the time stamps in TrueHeave files to the project.
7. If you are loading files logged over the GPS week number rollover, use the drop-down box to select a reference date for the files to be loaded.
8. To apply the TrueHeave data to the track line, click **Load**.

Load Attitude/Navigation Data

Load attitude and navigation data from Applanix POS MV or POSProc files to selected track lines.

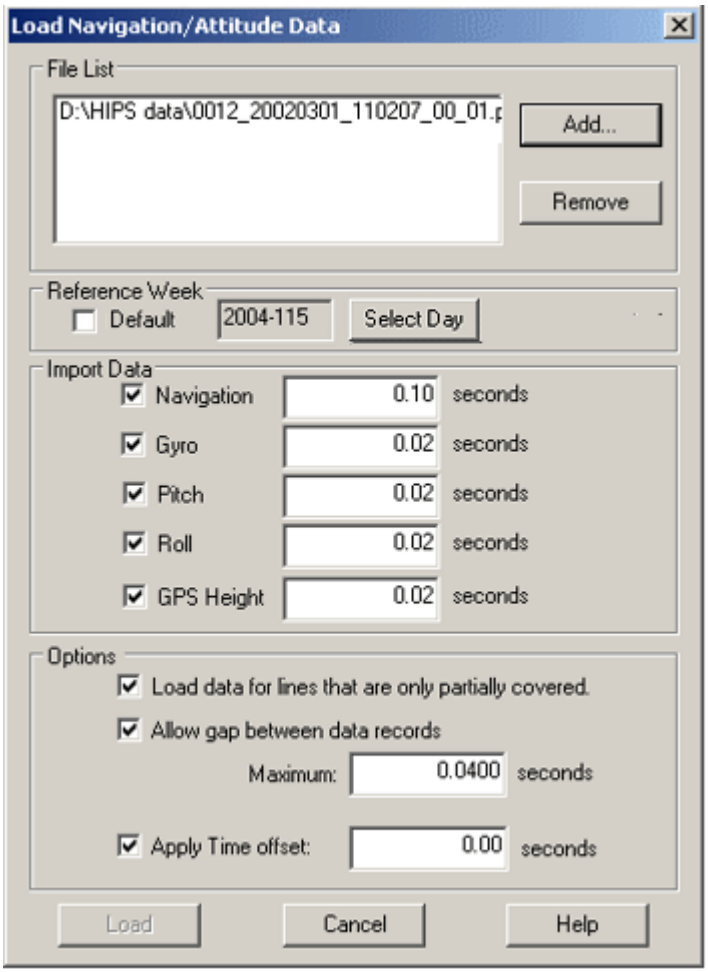
1. Select one or more track lines.

Attitude/Navigation

2. Select the Attitude/Navigation command.

Menu	Process > Load Attitude/Navigation Data

The Load Attitude/Navigation Data dialog box is displayed.



3. Click **Add** to select the files by browsing to their location. You can load either POS MV files OR load POSProc files, but not both at the same time.
4. To remove a file from the *File List*, select the file and click **Remove.**

Reference Week

5. [Optional] To accommodate a GPS week rollover event, you can select the Julian date of the day the data files were created. The rollover cannot span more than 7 days.

Import Data

6. Click the appropriate check boxes to choose which attitude/navigation records you want to import into the track line(s) from these files.

7. To reduce the volume of large attitude records to a manageable level, enter an output interval (in seconds).
8. The *Apply to all lines that are partially covered* option will load the navigation data even if the data does not cover the entire line.
9. Select the *Allow gap between records* check box to limit the interval allowed between consecutive data records in the file. If an interval exceeds the amount specified in the *Maximum* field, the data will not be loaded and the process will stop.
10. If the time stamp in the file does not match the time stamps in the project, you select the *Apply Time Offset* option and type a time value (in seconds).
11. Click **Load**.

Sound Velocity Processing

HIPS can perform post-processing of sound velocity profiles (SVP) to raw travel time and angle data. The result is a re-computation of the across-track/along-track/depth-coordinate triplet for soundings as seen in the Swath Editor and Single Beam Editor. For effective sound velocity correction, the SVP section of the HIPS Vessel file must be defined. (see "SVP" ON PAGE 142)

It is recommended that sound velocity correction be executed before cleaning the data. This is because the process applies the vessel motion data to the bathymetry, which makes cleaning the data easier in the interactive editors.

Sound velocity processing is not available for all types of multibeam and single beam sonar data. Only those sonar formats that provide raw travel time and angle from the transducer are currently supported. This includes multibeam bathymetry from the XTF, XSE and Hypack formats.

For all other multibeam sonar formats, it is assumed that sound velocity corrections were applied during data acquisition.

CARIS HIPS 6.1 provides sound velocity corrections for Simrad EM format data.

Single beam bathymetry converted into HIPS using the HIPS Generic Data Parser (see "INTRODUCTION" ON PAGE 628) can recover the raw travel time data. This allows you to apply sound velocity profiles to the single beam data in HIPS.

Sound velocity information converted to HIPS format can be viewed and modified in the SVP Editor. (see "SOUND VELOCITY EDITOR" ON PAGE 287)

SVP Files

Sound Velocity Profile files are in text format and can contain more than one profile. SVP files are similar to one shown below.

```
[SVP_VERSION_2]
E:\HIPS\Svp\sampleSvp_v2.svp
Section 2000-198 22:37:00 47:15:04 -54:19:51
   1.00 1405.00
   5.00 1440.00
  10.00 1456.00
  15.00 1480.00
  20.00 1490.00
  25.00 1494.00
  40.00 1500.00
  70.00 1505.00
  130.00 1510.00
Section 2000-198 23:00:00 47:15:03 -054:19:52
   1.00 1408.00
   5.00 1430.30
  10.00 1440.56
  15.00 1456.00
  20.00 1475.00
  25.00 1485.00
  40.00 1500.00
  70.00 1550.00
```

The SVP file has the following characteristics (starting from the top):

- The section labelled [SVP_VERSION_2] specifies the version of the SVP file.
- The path and name of the file.
- A section heading for each profile. The section heading contains:
  - the year and day (Julian date) that the profile was recorded
  - time that the profile was recorded
  - the latitude and longitude of the profile location.
- A list of depth/speed values.

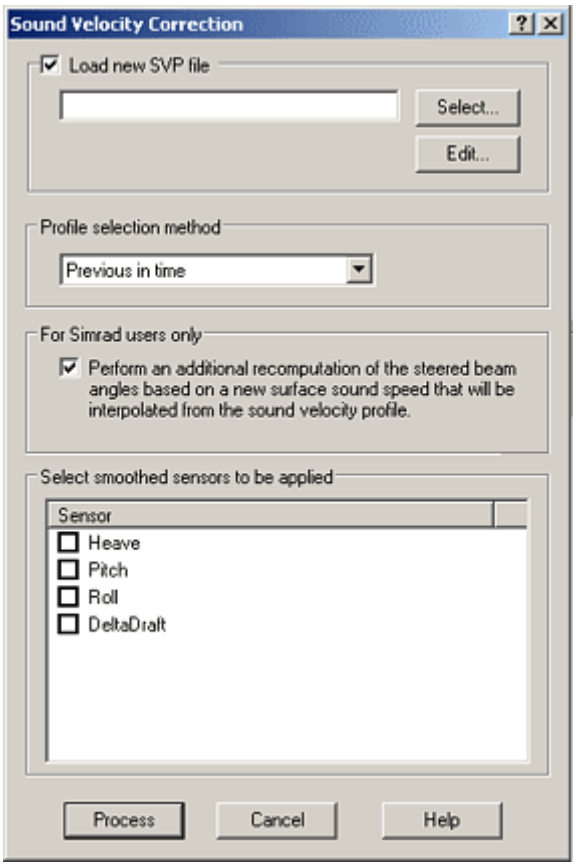
Sound Velocity Correction

The sound velocity correction process uses a ray tracing algorithm to apply the sound velocity profiles. This means that the SVP section of the vessel configuration be defined. You can use either regular attitude data or smoothed data for applying vessel motion data to the ray-tracing algorithm (see "SMOOTH DATA" ON PAGE 278).

1. Select a track line or group of track lines.
2. Select the Sound Velocity Correction command.

Menu	Process > Sound Velocity Correction
Tool	

The Sound Velocity Correction dialog box is displayed.



3. Select the *Load New SVP File* check box.
4. Click **Select** to choose an SVP file.

The Open an SVP File dialog box is displayed.

5. Select the file you want, or type the file path in the File Name text box. The default directory is …\Hips\Svp.
6. Click **Open**.

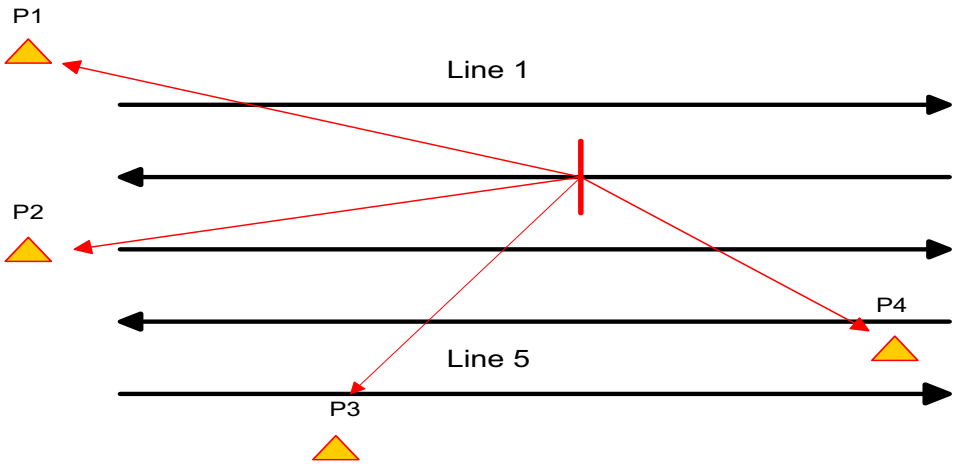
The file name and path is displayed in the text box.

7. If you want to edit the data with the SVP editor, click **Edit** (see "SOUND VELOCITY EDITOR" ON PAGE 287). .

When there is more than one sound velocity profile, the program can use one of four options for selecting an SVP at the time of each swath:

- *Previous in Time*: Select the profile with a time stamp prior and nearest to the time of the swath.
- *Nearest in Time*: Select the profile with a time stamp nearest to the time of the swath. This can either be prior to or after the time.
- *Nearest in Distance*: Select the profile with a position nearest the position of the swath. This requires that positions be included in the SVP file.
- *Nearest Distance Within Time*: Select the profile with a position nearest to the swath and has a timestamp nearest the time of the swath within a selected range.

The following diagram demonstrates the relationship of the four SVP options to the swath.

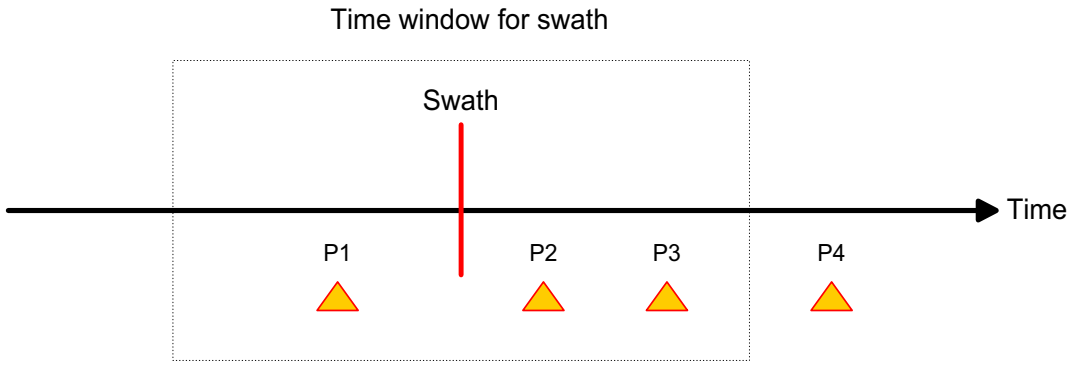


P1=Previous in Time

P2=Nearest in Time

P3=Nearest Distance Within Time

P4=Nearest in Distance



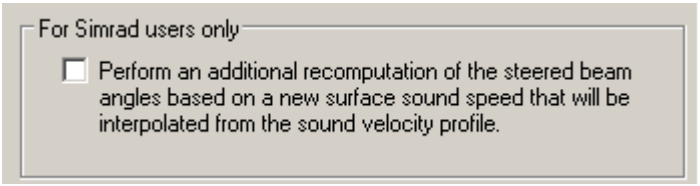
The above two graphics demonstrate the relationship of the profile options to the time/location of the swath. P1, P2 and P3 all are calculated within a time period. P4 is calculated independently of the time period.

8. Select one of above options by clicking the appropriate check box. If you select the *Nearest in Distance and Time* option, make sure you select a time range for the timestamp.

Simrad EM data

For Simrad EM data, the Sound Velocity Correction function requires the range/angle datagram to be recorded. The function, at a minimum, provides a re-computation of the across-track/along-track/observed depth solution based on the new sound velocity profile starting with the recorded range/angle data for each beam.

An additional option switch is provided on the dialog box to perform an optional re-computation of the steered beam angles when you believe the recorded surface sound velocity is not reliable. In this case, a new surface sound velocity will be interpolated for each swath from the new sound velocity profile.


For Simrad users only
Perform an additional recomputation of the steered beam angles based on a new surface sound speed that will be interpolated from the sound velocity profile.

9. Select this check box to perform an additional re-computation of the steered beam angles based on a new surface sound speed that is interpolated from the sound velocity profile.
10. Select which smoothed sensor data is to be applied during sound velocity correction. Sensor data can be smoothed in the Attitude Editor (see "SMOOTH DATA" ON PAGE 278).
11. Click **Process** to apply the SVP file to the selected data.

View SVP positions

To see the positions of the sound velocity profiles in the Display window.

Menu	View > Display SVP Positions

1. Select the Display SVP Positions command.

The Open SVP File dialog box is displayed.

2. Select a file and click **Open**.

The SVP symbols are displayed in the Display window. The symbols use the CARIS feature code selected in the Options dialog, and text of the SVP date and time.

An SVP Positions data layer is displayed in the Session layer tree:

Menu	View > Close SVP Positions

1. To remove the SVP layers from the display, select the Close SVP Positions command.

Navigation

Navigation data can be examined and cleaned for positioning outliers. This should be performed before merging to produce final position and depth files. However, you are not required to examine navigation data.

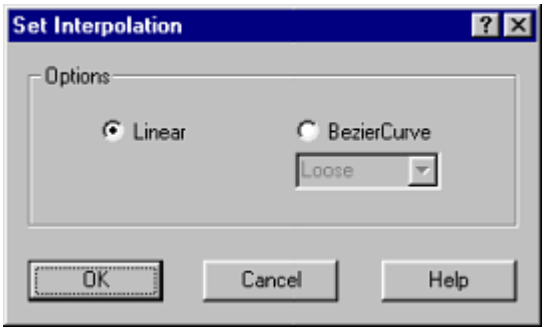
For more information, see "NAVIGATION EDITOR" ON PAGE 261.

The default interpolation method for navigation data is linear. However, you can optionally define one of three levels of Bezier Curve fitting.

1. Select a line.
2. Select the Set Interpolation command.

Menu	Tools > Set Interpolation

The Set Interpolation dialog box is displayed.



3. Select an interpolation option: *Linear* or *Bezier Curve* (see "LINE INTERPOLATION" ON PAGE 265).
4. Click **OK.**

Swath Cleaning

During data conversion, the original data file is separated into a number of distinct files, each storing a different types of information. All are related by time. One of these files is the Observed Depths file. This file contains measured single beam, sweep, or swath soundings as reported by the data acquisition system.

The Swath Editor displays the contents of the file for sweep and swath systems on the screen in graphic form and lets you to perform cleaning and filtering operations on the data.

For more information on the Swath Editor see "SWATH EDITOR" ON PAGE 297.

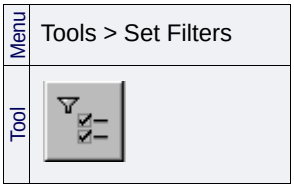
Swath Filtering

When dealing with high-volume bathymetric datasets, it is advantageous to be able to reject outliers automatically. To do this, HIPS has a set of swath filters. There are three basic types of filters:

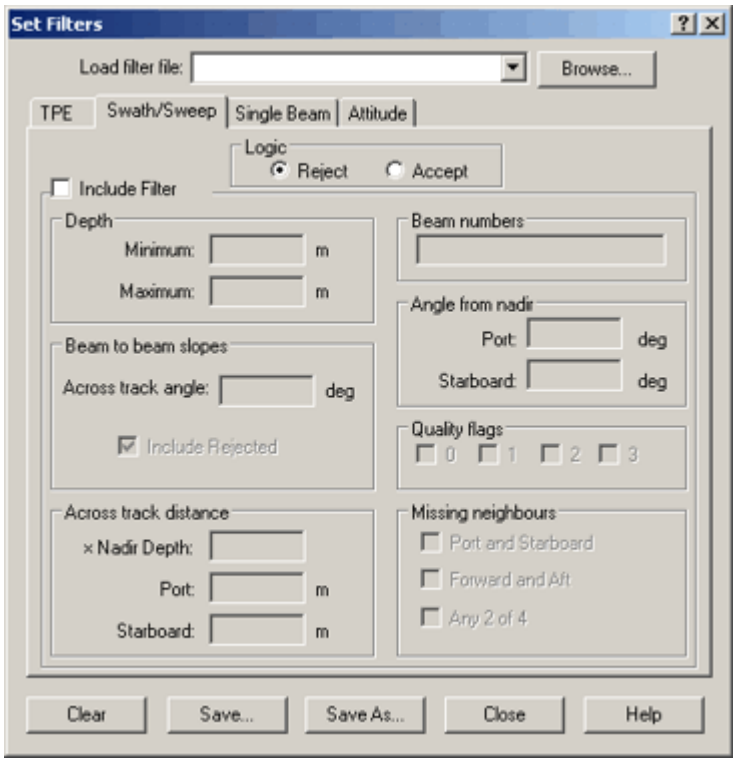
- **Spike detection**: The minimum and maximum depth filters and the beam-to-beam slope filters.
- **Swath reduction**: Across-track distance filters and nadir angle filters.
- **Sonar quality**: Quality values set by the sonar and missing neighbours.

The parameters for filtering are set through the Set Filters dialog box. These parameters can be saved in a HIPS Filter File (.hff) that can be reopened in the Set Filters dialog box for use in another track line.

1. Select the Set Filters command.



The Swath/Sweep tab of the Set Filters dialog box is displayed



1. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

If you chose to load a filter file, all the values used in that file are shown in the fields. You can filter data or change any of the values in the fields.

2. Set the logic to *Accept* or *Reject*.
3. Select the *Include Filter* check box to activate the filtering options.
4. Select the filter parameters used to flag data (see below).

You can define and apply more than one filter test. The filters are applied in sequence. Any data rejected by one filter is not considered in the next filter test.

5. Click **Close**, or click **Clear** to remove data from the fields.
6. Click **Save As** to save the settings to a filter file.

The following filters are available.

- *Minimum Depth*.Reject any soundings that are shallower than the depth specified (e.g., 10 m).
- *Maximum Depth*. Reject any soundings that are deeper than the depth specified (e.g., 50 m).
- *Across Track*. Beam-to-beam slopes, across-track. For each beam on the same swath, calculate the slopes in degrees to the prior and post beams, and if both slopes exceed the defined value and are of opposite sign then reject the beam.

The *Include Rejected* option includes previously rejected soundings when recalculating the slopes during multiple runs of the filter.

- *Across track distance: X Nadir Depth*. Any soundings with an across-track distance, from the centre beam, greater than the nadir depth times this number (e.g,. 3.0) are rejected. For example, in 30 metres of water, with a setting of 3.0, all soundings with an across-track distance greater than 90.0 metres are rejected.
- *Across track distance: Port*. Filter a sounding if the across track distance on the port side is greater than the value entered in this field.
- *Across track distance: Starboard*. Filter a sounding if the across track distance on the starboard side is greater than the value entered in this field.
- *Beam Numbers*. Specify beams to be rejected. Separate individual beams with a space and specify a range of beams with a hyphen. For example, entering 12 32 56-60 72 rejects beams 12, 32, 56, 57, 58, 59, 60, and 72.
- *Angles from Nadir: Port and Starboard*. Filter beams that are outside of the designated angles (e.g., 60 degrees). Angles are computed from the nadir using depths and across-track distances (roll corrected).
- *Quality Flags*. If your system attaches quality codes to soundings, and those codes are transferred to HIPS during conversion, a sounding can be rejected according to its value.

Quality flags are currently supplied by Reson, Elac, Hypack, and GSF multibeam systems/formats.

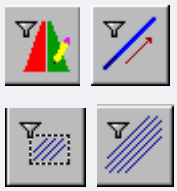
Missing neighbours are beams rejected during the conversion process into HIPS. This occurs if the beam is disabled in the vessel configuration or if the sonar system flags the beam as a bad detect in the raw data file.

- *Port-Starboard*. Beams are rejected if beams on each side of the swath are rejected.
- *Forward-Aft*. Beams are rejected if the adjacent beams in the previous and next swath are rejected.
- *Any 2 of 4*. Beams are rejected if two of the four neighbouring beams are rejected.

During filtering, only accepted soundings are used; therefore, the soundings rejected after one pass are not considered in the next.

Apply Filters

The last step is applying the automatic depth filter to the swath data.

Menu	Tool > Apply Filters > 1 Screen/To End of Line/Selected Lines/ All Lines
Tool	

1. Select an Apply Filter command to apply the criteria set by the Set Filters dialog box.

You have four options for applying the filters.

- **Filter 1 Screen**: Apply the filters only to the swaths that are currently visible in the Plan View of the Swath Editor.
- **To End of Line**. Apply the filters to the track line currently open in the Swath Editor, from the first swath currently visible in the Plan view to the end of the line.
- **Selected Lines**. Apply the filters to all of the currently selected track lines. This function does not require that the Swath Editor be open.
- **All Lines**. Apply the filters to all track lines currently open in the Display window. This function does not require that the Swath Editor be open.

Total Propagated Error

Total Propagated Error (TPE) is used to assign a horizontal error estimate (HzTPE) and a depth error estimate (DpTPE) to each sounding.

TPE is essential for the following functions:

- creating S-44 or S-57 compliant datasets (see "TPE Filtering" on page 236)
- calculating Bathymetry Associated with Statistical Error (BASE) Surfaces weighted by uncertainty (see "Uncertainty Weight" on page 405)

TPE is derived from a combination of all individual error sources. The following errors (among others) contribute to TPE:

- nav/gyro/heave/pitch/roll/tide errors
- latency error estimate
- sensor offset error estimates
- individual sonar model characteristics

Error values are located in the vessel file for the survey and the DeviceModels.xml file in …\HIPS\System. Other errors are directly entered in the Compute TPE dialog box.

The following sonar error-models in the DeviceModels.xml have been tested by the University of New Hampshire (UNH).

- Simrad EM300
- Simrad EM1000
- Simrad EM1002
- Simrad EM3000
- Simrad EM3000D
- Reson SeaBat 8101
- Reson SeaBat 9001
- Reson SeaBat 9003
- Reson SeaBat 8125
- Atlas Hydrosweep DS
- Elac Nautik 1180
- Seabeam 2112

Other sonars listed in the file may make assumptions that could produce less rigorous results than the error models supplied by UNH.

Compute TPE

The Compute TPE (Total Propagated Error) command applies HzTPE and DpTPE values to each sounding along the track line.

The HzTPE and DpTPE values are stored line-by-line with indexing to profiles and beams. You can find specific error values for individual soundings by querying the soundings in the Swath, Single Beam, and Subset Editors. The HzTPE and DpTPE values are displayed in the Query View.

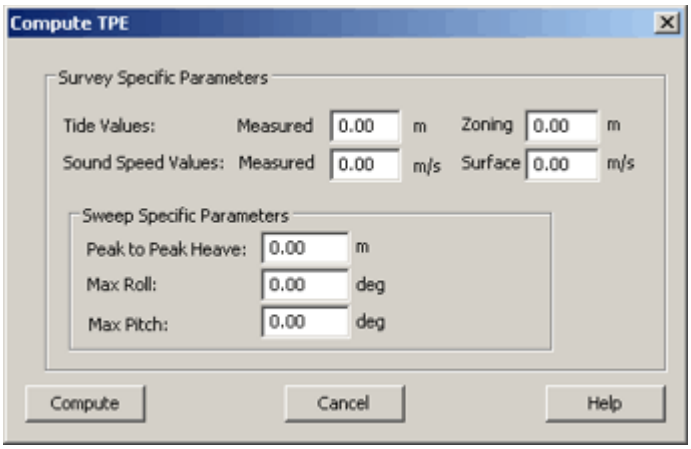
Static errors (such as those related to sensor position) that are used to compute the TPE are read directly from the project's vessel file and the DeviceModels.xml file. These errors are directly entered in the TPE computation.

Dynamic errors due to tide and sound velocity values, which can change from time to time, due to a variety of reasons, are entered via the dialogue box, to make it easy for the user to change them.

1. Select a track line in the Display window so it is highlighted.
2. Select the Compute TPE command.

Menu	Process > Compute TPE
Tool	

The Compute TPE dialog box is displayed.



3. In the *Tide Values* fields, enter the error offsets to be applied to all lines being processed.
   - The *Measured* offset is the error value for the tide station.
   - The *Zoning* offset is the error value in range calculation for a tide zone file.
4. Enter error offsets for *Sound Speed Values*.
   - The *Measured* offset value is to compensate for inaccuracies in SVP measurements.
   - The *Surface* offset value is used to account for errors in surface sound speed measurements.

Transducer motion offsets for sweep surveys are entered in the following three fields.

5. Type the estimated observed heave offset in the Peak-to-Peak Heave field.
6. Type the roll values for the transducers in the *Max Roll* field.
7. Type the pitch values for the transducers in the *Max Pitch* field.
8. Click **Compute**.

The results are displayed in the Output tab of the Worksheet window. A log file is also created showing the results.

You should update TPE if the observed depths have been modified.

TPE Filtering

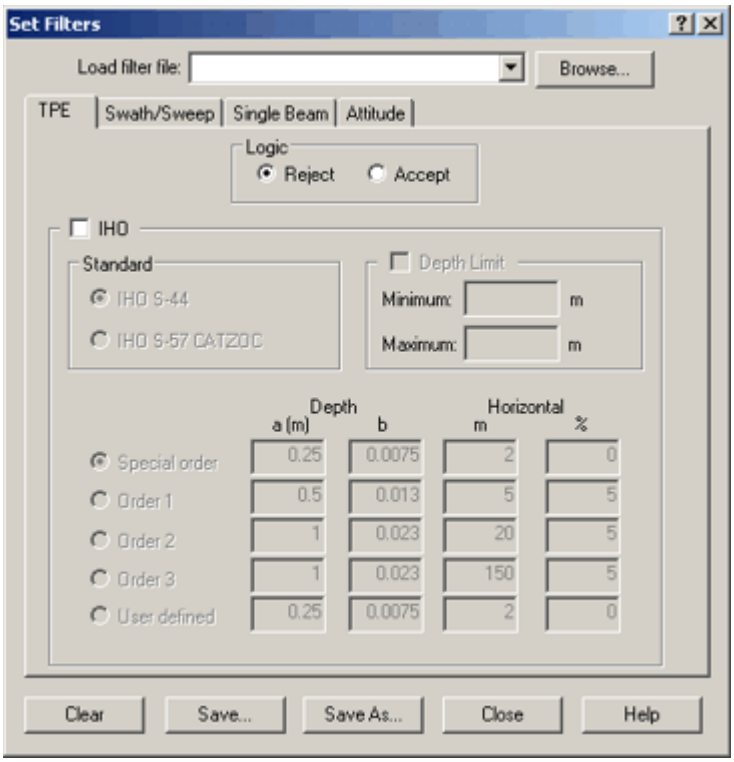
The TPE filtering option rejects (or accepts) soundings with uncertainty values that fall outside limits set by International Hydrographic Organization (IHO) standards.

These parameters can be saved in a HIPS Filter File (.hff) that can be reopened for use on another track line.

1. Select a track line.
2. Select the Set Filters command.

Menu	Tools > Set Filters
Tool	

The TPE tab of the Set Filters dialog box is displayed.



3. Select the TPE tab.

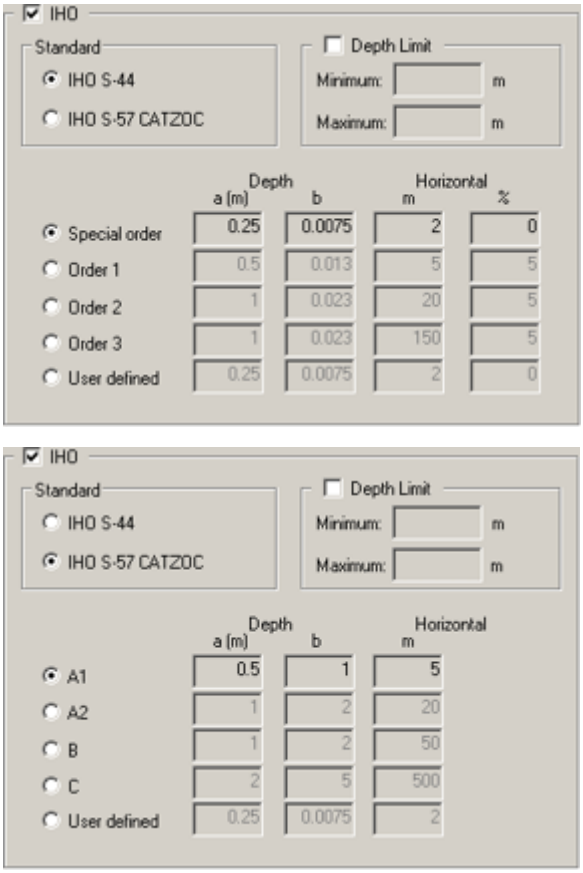
4. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

If you load a filter file, all the values used in that file are shown in the fields. You can change any of the values.

5. Select either a *Reject* or *Accept* logic command to apply to the filtered data.
6. Complete all data in the necessary fields (see below).
7. Click **Close**, or click **Clear** to remove data from the fields.
8. Click **Save As** to save the settings to a filter file.
9. Apply the filters to the data.

The filter compares each sounding's HzTPE and DpTPE values against the depth and horizontal error limits for a specific S-44 survey order or S-57 zone of confidence (CATZOC) attribute. All soundings with HzTPE and DpTPE values outside those limits are rejected (or accepted).

When you select an S-44 or S-57 CATZOC standard, the dialog box is refreshed to display the options associated with that standard. The options for the IHO S-44 survey orders and S-57 CATZOC attribute are shown below.



Each survey order or zone of confidence contains the following fields:

- *a*: constant depth error (i.e. the sum of all constant errors)
- *b*: factor of the depth dependent error
- *m*: position limit for horizontal errors
- %: percentage of depth used to calculate horizontal error (S-44 only).

Depth error limit

To calculate the error limits for depth accuracy, the constant depth error (a) and the factor of the depth dependent error (b) are combined with depth (d) and the depth dependent error (b * d) in the following formula:

$$\text{Error limit for depth accuracy} = \pm\sqrt{a^2 + (b \times d)}^2$$

When the filter is run, soundings with a vertical uncertainty outside the error limit for the survey order or zone of confidence are rejected (or accepted).

Horizontal error limit

To calculate horizontal accuracy, the limit for horizontal error (m) and (if applicable) the depth percentage (%) for the selected survey order or zone of confidence are compared against

a sounding's HzTPE. Soundings with a horizontal uncertainty outside the limits are rejected (or accepted).

Run Filter

1. Select either the *ISO 2-44* or *S-57 CATZOC* options.

The IHO frame of the TPE tab is refreshed to display the S-44 survey order options or the zones of confidence (see above).

You can apply filtering to a selected depth range by entering the maximum and minimum soundings for that range. Leave these fields empty to apply filtering to every sounding along the track line.

2. Select the *Depth Limit* check box to apply the filters to a depth range.
3. Type depth levels in the *Minimum* and *Maximum* fields to set a range for the filters.
4. Select a survey order or zone of confidence option.

All fields in the *User defined* option are activated when this option is selected.

5. Type *Depth* and *Horizontal* values.
6. Run one of the Filtering commands.

Menu	Tools > Apply Filters > Selected Lines/All Lines
Tool	

Load Error Data

Override error values in the HIPS Vessel File with real-time error values.

Applanix POS MV files and Applanix POSProc files contain real-time attitude and navigation error values recorded during the survey. This means that different error values are recorded for different conditions. This results in more precise error estimates than the static error values recorded in the HIPS Vessel File.

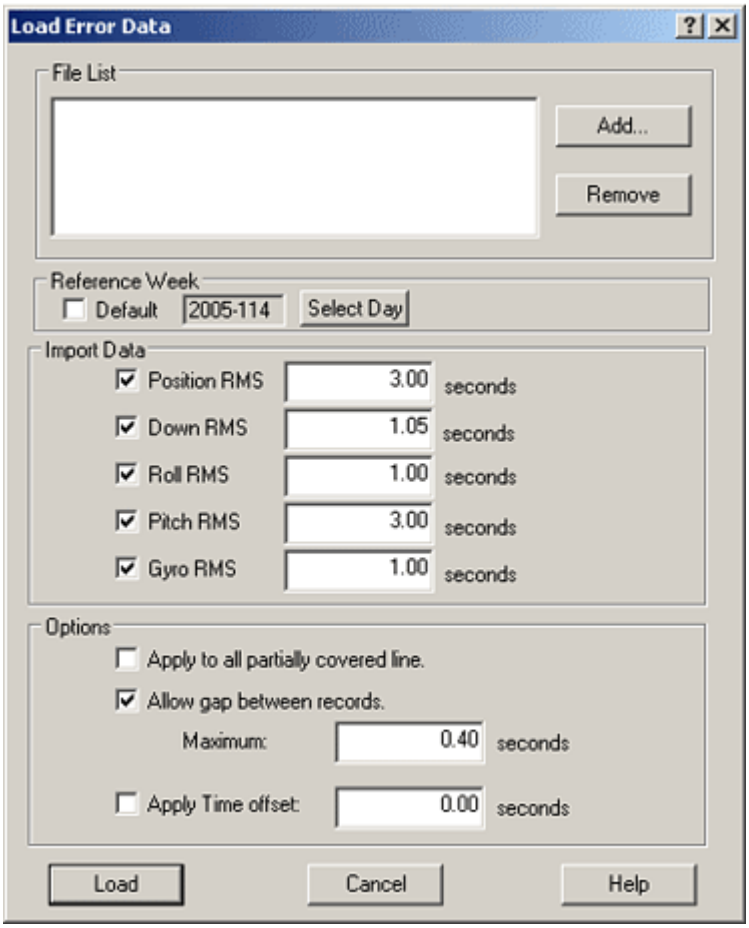
The files must be timestamped with valid GPS entries to determine the time of the data.

The imported files must contain the Julian date of the day the data was acquired in the form: YYYY-JJJ-filename or filename YYYY-JJJ

1. Select a track line in the Display window.
2. Select the command.

Menu	Process > Load Error Data

The Load Error Data dialog box is displayed.



3. Click **Add** to select files. You can load either POS MV files OR load POSProc files, but not both at the same time.

4. To remove a file from the *File List* field, select the file and click **Remove**.

Reference Week

5. [Optional] To accommodate a GPS week rollover event, you can select the Julian date of the day the data files were created. The rollover cannot span more than 7 days.

Import Data

6. Choose which error records in the Applanix files you want to import into the line by clicking on the appropriate check boxes.
7. To reduce the volume of large attitude records to a manageable level, enter an output interval (in seconds).

Options

8. The *Apply to all partially covered line* option will load the error data even if the data does not cover the entire line.
9. Select the *Allow gap between records* check box to limit the interval allowed between consecutive data records in the file. If an interval exceeds the amount specified in the *Maximum* field, the data will not be loaded and the process will stop.
10. If the time stamp does not match the time stamps in the project, you can select the *Apply Time offset* option and enter a time value (in seconds).
11. Click **Load**.

Update Backscatter

This function updates the backscatter imagery for a line after swath cleaning is complete. This is because the backscatter data is referenced directly to the individual beams of the bathymetry. Thus, after soundings have been rejected, this function removes their intensity values from the backscatter image.

1. Select a track line.
2. Select the Update Backscatter command.

Menu	Tools > Update Backscatter

Single Beam Cleaning

Using the Single Beam Editor you can perform cleaning and filtering on single beam data converted from single and dual frequency echo sounders. This tool inherits some of the functions of the Swath Editor such as vertical exaggeration and filtering functions.

For more information about the Single Beam Editor see "SINGLE BEAM EDITOR" ON PAGE 325.

Single Beam Filtering

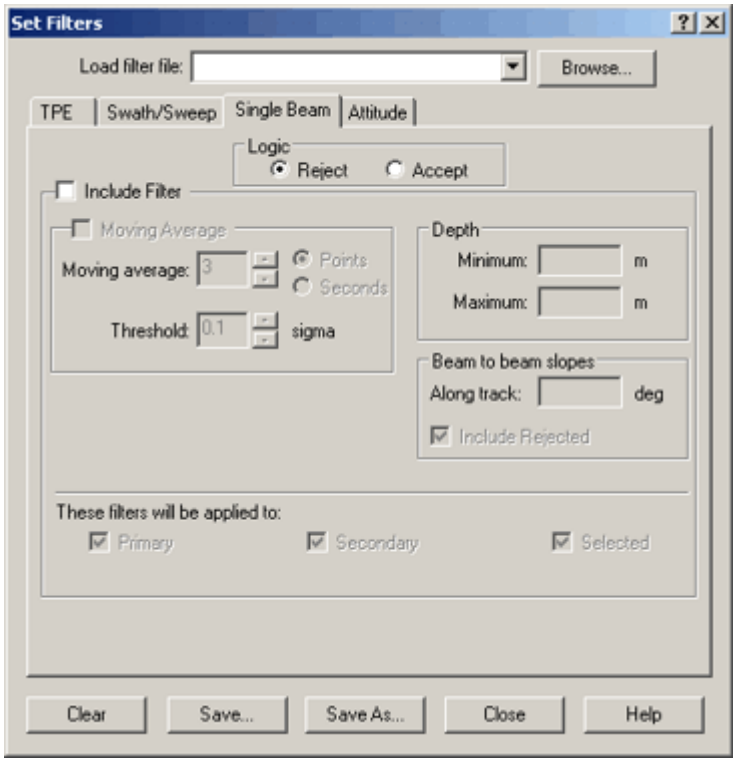
When dealing with bathymetric datasets, it is advantageous to reject outliers automatically. To do this, the Single Beam Editor has a set of filters.

The parameters for these filters are set through the Set Filters dialog box. The parameters can be saved in a HIPS Filter File (.hff) that can be reopened for use in another track line.

1. Select a track line.
2. Select the Set Filters command.

Menu	Tools > Set Filters
Tool	

The Single Beam Filters tab of the Set Filters dialog box is displayed.



3. Make sure the Single Beam tab is displayed.
4. [Optional] Select an existing HIPS Filter File from the list or click **Browse** to select a file.

If you chose to load a filter file, all values in that file are shown in the fields. You can go to Step 7 or change any of the values.

5. Select either a *Reject* or *Accept* logic command to apply to the filtered data.
6. Select the *Include Filter* check box to activate the filtering options.
7. Select from three filtering options:
   - *Moving Average* (see "Moving Average Filter" on page 245)
   - *Depth* (see "Depth" on page 246)
   - *Beam-to-beam slopes* (see "Beam-to-Beam Slopes" on page 246)

You can define and apply more than one filter test. The filters are applied in sequence. Any data rejected by one filter is not considered in the next filter test.

8. Select the frequency to apply to the filters by making sure the appropriate box is checked.
9. Click **Close**, or click **Clear** to remove data from the fields.
10. Click **Save As** to save the settings to a filter file.

Moving Average Filter

The Moving Average Filter rejects (or accepts) data with difference values that fall outside of defined threshold limits. To view the Moving Average filtering options in the Single Beam Editor, see "DISPLAY OPTIONS" ON PAGE 327.

A moving average is calculated for each sounding (data point) by using the adjacent soundings. The amount of adjacent data is specified either as the number of points or as a time period.

1. Select the *Moving Average* check box to enable the associated fields.

Set the moving average using seconds

This option uses a time range (in seconds) to calculate a mean average for each data point.

For example, if you select 10 seconds as the moving average, the filter calculates a mean average for any data point using itself and any data points in the five-second range either side of it.

Setting the moving average using points

This option uses a range of adjacent data points to calculate a mean average for each data point.

For example, if the moving average is set to five data points. A mean for any data point is calculated using itself and the two data points on either side of it (if available).

2. Select how you want the moving average to be calculated by selecting either *Points* or *Seconds*.
3. Use the arrow buttons to select the number of seconds or data points.

A difference value for each sounding is calculated for each sounding from the adjacent soundings.

Thresholds

The thresholds indicate what difference values are rejected (or accepted) during filtering.

The upper threshold is determined by the following formula:

```
original mean value + (threshold value X the
standard deviation)
```

The lower threshold is determined by the following formula:

```
original mean value – (threshold value X the
standard deviation)
```

4. Select a threshold value by clicking the arrow buttons.

Depth

1. Enter the depth values for filtering soundings. There are two options:
   - *Minimum Depth*: Reject (or accept) any soundings that are shallower than the depth specified (e.g., 10 m).
   - *Maximum Depth*: Reject (or accept) any soundings that are deeper than the depth specified (e.g., 50 m).

Beam-to-Beam Slopes

For each sounding, calculate the slopes, in degrees, to the previous and next soundings. If both slopes exceed the defined value and are of opposite sign, then reject the sounding.

1. Enter the degree value for the slope.
2. Select the I*nclude Rejected* option to include previously rejected soundings when recalculating the slopes during multiple runs of the filter.

Apply Filters

The last step is applying the automatic depth filter to the data.

1. Select a Filter command to apply the criteria set by the Set Filters dialog box.

Menu	Tool > Apply Filters > 1 Screen/To End of Line/Selected Lines/ All Lines
Tool	

You have four options for applying the filters.

Filter 1 Screen. Apply the filters only to the soundings that are currently visible in the Single Beam Editor.

To End of Line. Apply the filters to the track line currently open in the Single Beam Editor, from the first sounding currently visible in the editor to the end of the line.

Selected Lines. Apply the filters to all of the currently selected track lines. This function does not require that the editor be open.

All Lines. Apply the filters to all track lines currently open in the Display window. This function does not require that the editor be open.

Restart Cleaning

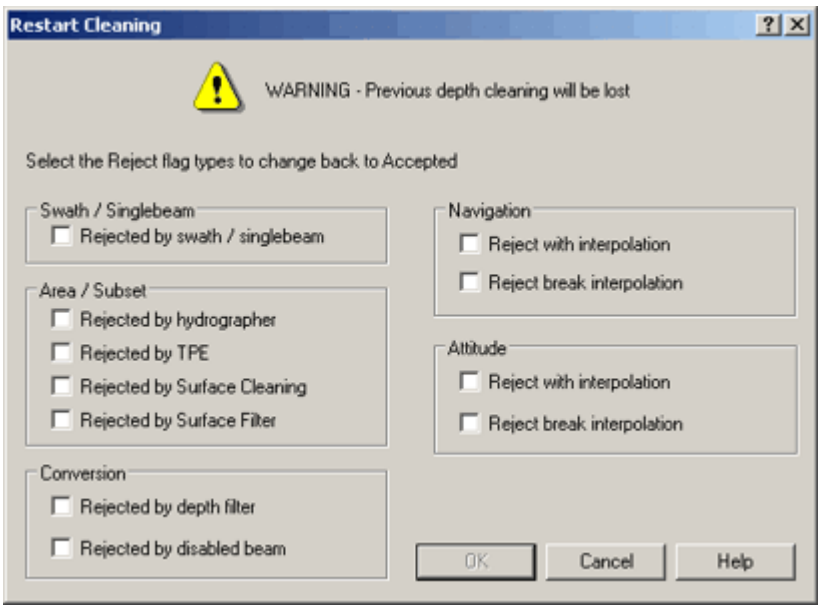
The Restart Cleaning function resets status flags from rejected to accepted for sounding, attitude, and navigation data.

> **Warning**: All previous depth cleaning is lost once this command is implemented.

1. Select the track line(s).
2. Select the Restart Cleaning command.

Menu	Edit > Restart Cleaning
Tool	

The Restart Cleaning dialog box is displayed.



3. Select any of the following check boxes to select a rejected data criteria.
   - *Rejected by Swath Editor/Single Beam Editor*: Reset to accepted sounding data that was rejected in the Swath or Single Beam Editors, or by the Swath or Single Beam filters.
   - *Rejected by Hydrographer*: Reset to accepted the sounding data that was interactively rejected in the Subset Editor.
   - *Rejected by TPE*: Reset to accepted the data that was rejected during Total Propagation Error (TPE) filtering.
   - *Rejected by Surface Cleaning*: Reset to accepted data that was rejected during Surface Cleaning.
   - *Rejected by Surface Filter*: Reset to accepted data that was rejected during CUBE filtering.
   - *Rejected by Depth Filter*: Reset to accepted data that was rejected by the depth filter during conversion.

- *Rejected by Disabled Beam*: Reset to accepted data that was rejected due to the disabled beam flag during conversion.
- *Rejected with Interpolation (Navigation)*: Reset to accepted navigation data that was rejected without interpolation.
- *Rejected break Interpolation (Navigation)*: Reset to accepted navigation data that was rejected without interpolation.
- *Rejected with Interpolation (Attitude)*: Reset to accepted attitude data that was rejected with interpolation.
- *Rejected break Interpolation (Attitude)*: Reset to accepted attitude data that was rejected without interpolation.

4. Click **OK**.

The selected data is now flagged as accepted.

Delta Draft

The Load Delta Draft function lets you import your model for dynamic changes in draft and have these changes applied to the soundings during the Merge process. Delta Draft represents a change in draft that is positive as draft increases and negative as draft decreases. The data, which is loaded from a ASCII text file, overrides the Dynamic Draft information in the vessel file. The Delta Draft value is interpolated between timestamps in Merge and added to the Observed Depth to obtain the Final Depth.

The input format for the Delta Draft text file is similar to the COWLIS tide format. Each record is in the following format:

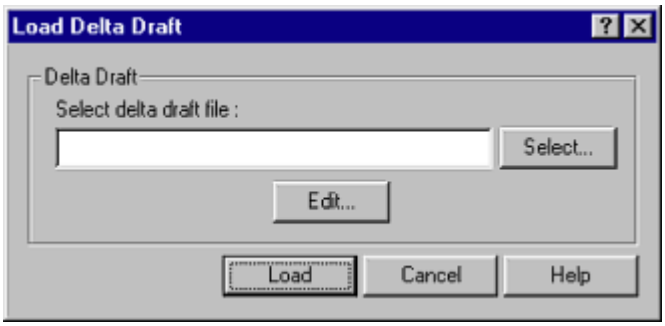
```
YYYY/MM/DD HH:MM:SS.SSS 1.234
```

The Delta Draft function can also be used to load recorded depth versus time data for an underwater platform on which the multibeam sonar is mounted, such as a ROV (Remote Operated Vehicle) or AUV (Autonomous Underwater Vehicle), or towed vehicle.

1. Select the track line(s).
2. Select the Load Delta Draft command.

Menu	Process > Load Delta Draft
Tool	

The Load Delta Draft dialog box is displayed.



3. Type the path of the file or click **Select** to locate the file.
4. To make changes to the file, click **Edit**.

The file is opened in the default text editor (such as Notepad). Make any changes and save the file.

5. Click **Load**.

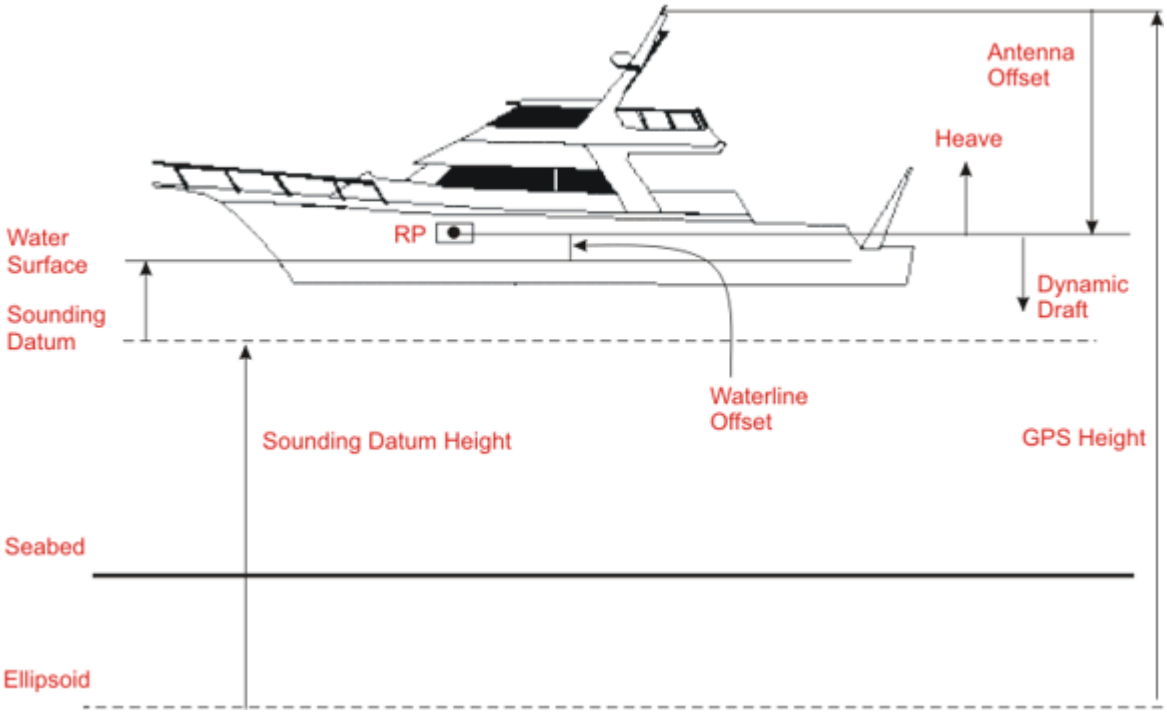
The delta draft is loaded into the line folder.

GPS Tide

The Compute GPS (Global Positioning System) Tide function provides an alternative to normal tidal observation for reducing soundings to the sounding datum

It is based on the recorded GPS ellipsoid height of the navigation antenna and uses the recorded vessel motion, if available, to reduce the ellipsoid height measurements to water level.

The following image demonstrates how GPS tide is calculated. The arrows indicate the positive sense of the values.



The full formula for GPS tide is shown below.

```
GPS Tide = GPS Height - Datum Height + Antenna
Offset - Heave + Dynamic Draft - Waterline Offset
```

The above diagram and equation show the complete formula for GPS tide where GPS Height needs to be motion corrected. However, in some systems, GPS Height may already be motion corrected. Applying sensor values (such as dynamic heave) again will only distort the final GPS Tide. The Compute GPS Tide command lets you include or omit selected sensor values.

There are two options for applying the sounding datum height(s) (distance from the ellipsoid to the sounding datum) to the track line.

- apply a single height to a track line, or
- apply a binary or ASCII text file containing a grid of sounding datum heights.

Two binary sounding datum models can be used for computation of GPS tide:

- GEOID99
- EGG97 (European Gravimetric Geoid 1997)

Alternatively, an ASCII sounding datum model can also be used. The data must be in the following format: Lat,Long,Z. Values are delimited using a comma.

An example of the file is shown below.

```
-33.848326,151.192435,20.4
-33.849484,151.192370,20.1
-33.849237,151.193173,19.7
-33.849506,151.194227,19.8
-33.848298,151.193916,20.0
-33.848402,151.193149,20.2
```

ASCII datum model files must follow a specific format:

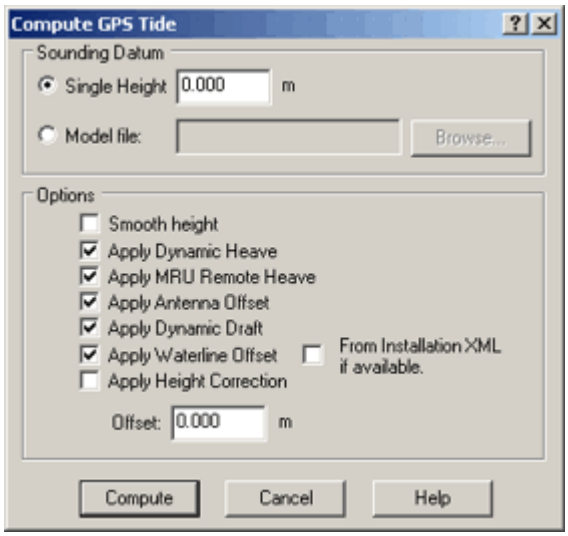
- Latitude and longitude must be in decimal degrees.
- The hyphen symbol (-) must be used to designate coordinates in the Southern or Western hemispheres.
- All files must be saved with a TXT or XYZ file extension.

To compute GPS tide, complete the following steps:

1. Select a track line or group of track lines.
2. Select the Compute GPS Tide command.

Menu	Process > Compute GPS Tide

The Compute GPS Tide dialog box is displayed.



3. Enter the *Sounding datum height* to apply a single height to a track line.
4. To apply a file containing a grid of sounding datum heights, type the path and name for the *Sounding Datum Model* file, or click **Browse** to select a file.
5. Select the *Smooth Height* check box to apply smoothed GPS heights to the track line (see "SMOOTH DATA" ON PAGE 278).

The following check boxes apply sensor values to GPS tide if GPS Height needs to be motion corrected.

6. Select *Apply Dynamic Heave* to apply dynamic heave (regular vessel heave, or true heave if exists).
7. Select *Apply MRU Remove Heave* to apply remote heave resulting from vessel roll/pitch on an offset mounted MRU.
8. Select *Apply Antenna Offset* to apply vertical offsets as a result of rotating GPS antenna by vessel roll/pitch.
9. Select *Apply Dynamic Draft* to apply dynamic draft (as a result of interpolating the draft table in HVF, or stored in HIPS as time series data).
10. Select *Apply Waterline Offset* to apply the waterline offset in the HIPS Vessel File (HVF).
11. Select *Apply Height Correction* to apply a static offset to the GPS antenna and type the value in the *Offset* field.
12. Select *From Installation XML* to apply the waterline value from the InstallationParameters.xml generated by the Simrad converter.
13. Click **Compute**.

The GPS Tide is calculated for the selected line(s). The settings in the dialog box are remembered by the application. You can open and view the GPS tide data in the Attitude Editor ("ATTITUDE EDITOR" ON PAGE 271).

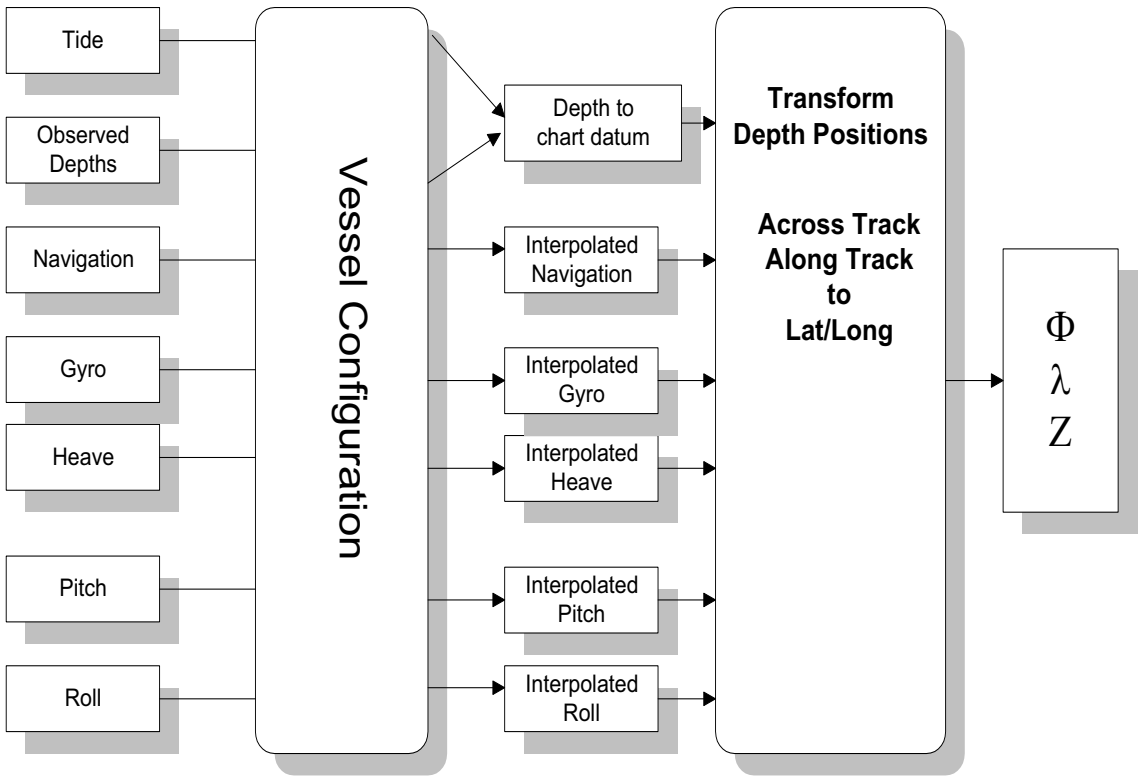
Merge

Lines must have tide loaded before they can be merged.

The following operations take place when merging:

- Recorded sensor information is compensated for constant time errors as noted in the vessel file.
- For each time-tagged depth record, the position for the centre of the swath profile is calculated based on the interpolated position fix at that time.
- For each depth record, the position of the sounding is calculated, based on the profile centre position and any gyro/heave/pitch/roll corrections that are appropriate (this is echosounder system dependent).
- Tide is interpolated and applied to each depth record.
- Draft is applied.

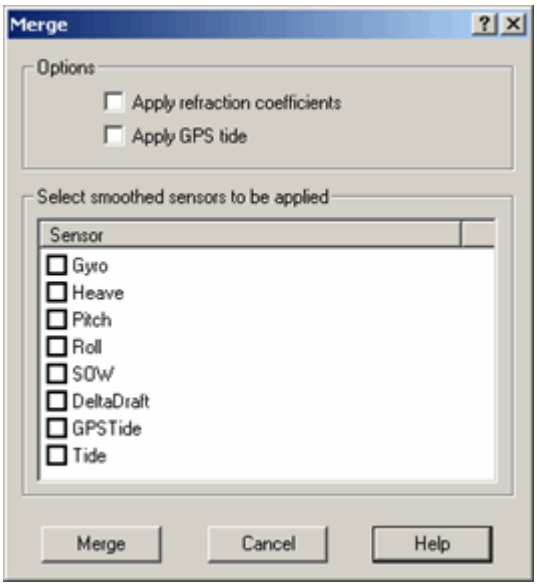
A processed depths file is created for each line. This file contains the final computed geographic position for each depth record.



Menu	Process > Merge
Tool	

1. Select a line or group of lines.
2. Select the Merge command.

The Merge dialog box is displayed.



3. Click the *Apply refraction coefficients* check box to apply refraction coefficients.
4. Click the *Apply GPS tide* check box to use GPS tide data instead of tidal observation data.

If you have a SmoothedCoefficients file (see "SMOOTH DATA" ON PAGE 278) for a sensor, you can apply this file during merge to smooth selected sensor data.

5. Click a check box beside a sensor to select the appropriate data that is to be smoothed during merge.
6. Click **Merge**.

You can view the progress of the merge process in the Worksheet window.

The merge is now complete. Other functions requiring georeferenced soundings can now be used such as the Subset Editor, Surface Cleaning, BASE Surfacing and others. The settings in the dialog box are remembered by the application.

Once a line is merged and changes are made to the sensor data used by merge, the track line is marked as Outdated. An Outdated track line must be remerged.

Quality Control

The Quality Control (QC) process in HIPS is typically used to statistically compare soundings recorded from check lines against selected attribute values from a BASE Surface or weighted grid in the same survey area.

Statistical variances between the check-line soundings and the attribute values are shown in a table in the Worksheet window once the QC tool is run.

1. Make sure that the Project tab of the Control window is visible.
2. Select a survey line. You can also, if needed, select a Project, Vessel, or Day folder.

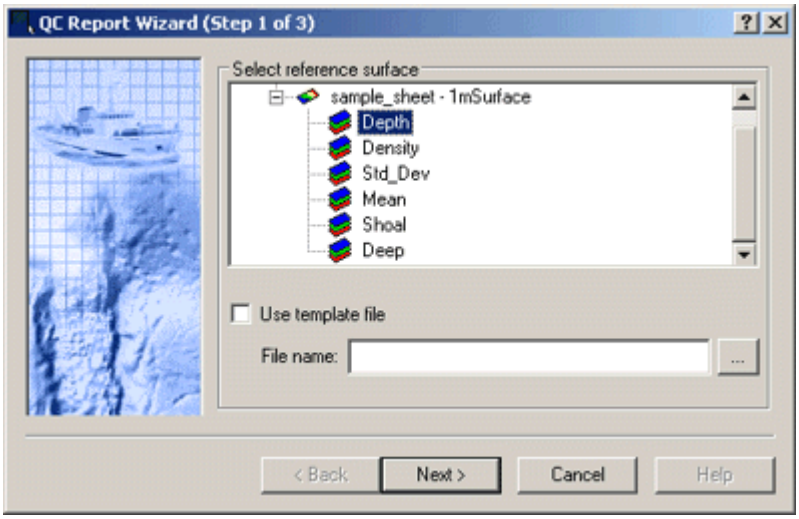
Quality Control

3. Select the Quality Control command.

Menu	Process > QC Report

QC Report - Step 1

The QC Report - Step 1 dialog box is displayed.



1. Select a BASE Surface attribute or weighted grid from the *Select Reference Surface* list.

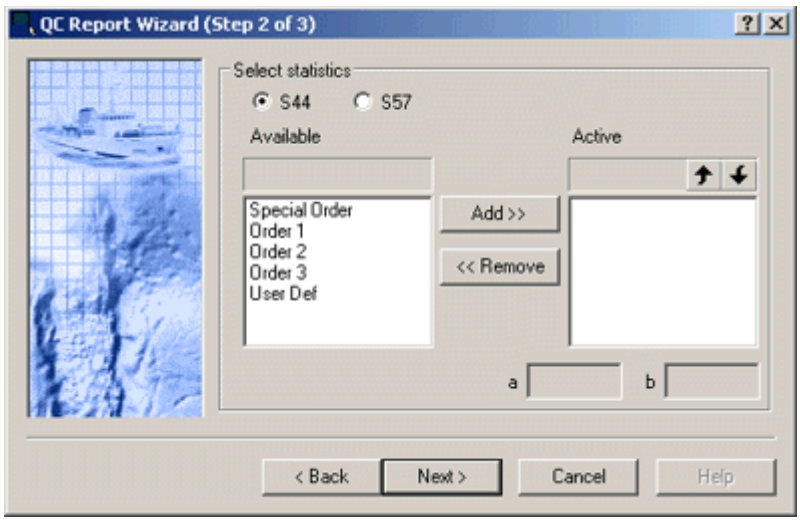
A template file is an XML file that contains output from the Quality Report. You can save the output as a template file or open an existing template file. If you open an existing template file, the settings saved in that file are applied to the options in the other dialog boxes.

2. Select the *Use template file* check box to create or open a template file.
3. Click **Browse** to select a location and name for the file; or select an existing file.

4. Click **Next**.

QC Report - Step 2

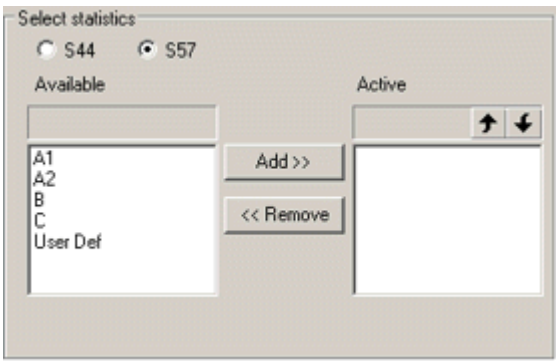
The QC Report - Step 2 dialog box is displayed.



This dialog box selects the IHO standard for determining what percentage of soundings fall within a selected error limit for depth accuracy. You can use the S-44 Ed. 4 survey orders or S-57 CATZOC zones of confidence. For more information on how error limit for depth accuracy is calculated, see "TPE FILTERING" ON PAGE 236).

1. Select either the *S-44* or *S-57* option.

The dialog box is refreshed to show the categories relevant to the selected option.



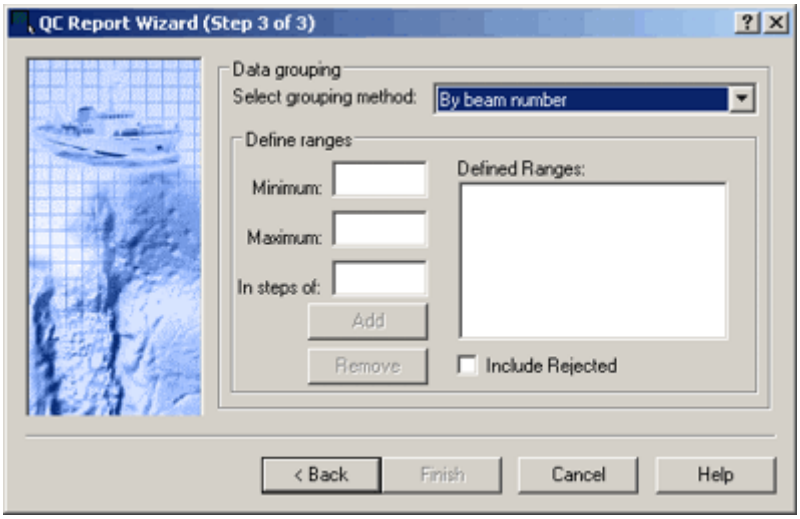
2. To display a category in the QC report, select it in the *Available* list and click **Add** to move to the *Active* list.

If you are using the user defined (*User Def*) category, the *a* (constant depth error) and *b* (factor of the depth dependent error) fields become active once it is moved to the *Active* list.

3. [Optional] Type the values for the user defined category in the *a* and *b* fields.
4. To remove a category, select it in the *Active* list and click **Remove** to move it back to the *Available* list.
5. Select a category and click the up or down arrow buttons at the top of the *Active* list to determine its order in the *Active* list. This will determine the order that data is displayed in the Worksheet window when the QC Report is generated.
6. Click **Next**.

QC Report - Step 3

The QC Report - Step 3 dialog box is displayed.



This dialog box defines how soundings are grouped in the QC Report. There are three options available:

- *Beam number*: Soundings are grouped according to across-track beam number.
- *Angle*: Soundings are grouped according to beam angle from nadir.
- *Distance*: Soundings are grouped according to across-track distance from nadir.

1. Select a method for grouping soundings from the drop-down list.

Soundings can be further organized by range. The range values must correspond to grouping method selected above.

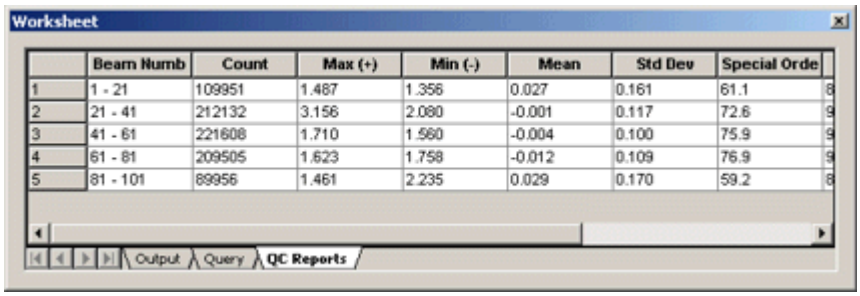
2. Type the range in the *Minimum* and *Maximum* fields.

Ranges are incremented according to the value entered in the *In steps of:* field. For example, for beam numbers 1 to 100 with an increment of 10, the beams are then displayed in the QC Report tab in 10 rows.

3. Type a value to increment the ranges.
4. Click **Add** to display the range in the *Defined Ranges* list.
5. To remove a range, select it in the *Defined Ranges* list and click **Remove**.
6. To include rejected soundings, select the *Include Rejected* check box.
7. Click **Finish** to complete the wizard and close the dialog box, or click **Cancel** to close the wizard without implementing changes.

Quality Control Statistics

The quality control report is displayed in the QC Reports tab of the Worksheet window.



The fields are explained in the following table.

Field	Definition
Beam Number	Range grouping as defined in the third dialog box in the wizard.
Count	Total soundings in range.
Max	Maximum distance of soundings above surface.
Min	Maximum distance of soundings below surface.
Mean	Mean difference of soundings to surface.
Std Dev	Standard deviation of mean differences is presented at one sigma or 68% Confidence Interval.
(Selected S-44 or S-57 categories)	Percentage of soundings that fall within the selected S-44 survey orders or S-57 CATZOC zones of confidence categories.

Side Scan Processes

A number of side scan processing options can be applied by selecting a track line without opening the Side Scan Editor. These processes can be used on one or more track lines.

If you use these commands, you should remember two important points:

- The process is applied to the entire line.
- Some of the signal graph options available in the Side Scan Editor are not available.

The following side scan processing options are available:

- Slant Range Correction (see "SLANT RANGE CORRECTION" ON PAGE 525)
- Beam Pattern Correction (see "BEAM PATTERN CORRECTION" ON PAGE 529)
- Angle-Varying Gain Correction (see "ANGLE-VARYING GAIN" ON PAGE 534)
- Time-Varying Gain Correction (see "TIME-VARYING GAIN" ON PAGE 535)
- Despeckle (see "DESPECKLE" ON PAGE 538
- Pure Rejected Contacts (see "EDITING CONTACTS" ON PAGE 542)
- SIPS Template Wizard (see "SIPS TEMPLATE WIZARD" ON PAGE 557)

Line Calibration

Open the HDCS (Hydrographic Data Cleaning System) to examine calibration lines for parameters such as transducer mounting offsets (pitch, roll, yaw), navigation latency or heave latency.

For more information about the Line Calibration function see "CALIBRATION" ON PAGE 481

9

Navigation Editor

Navigation Editor Interface

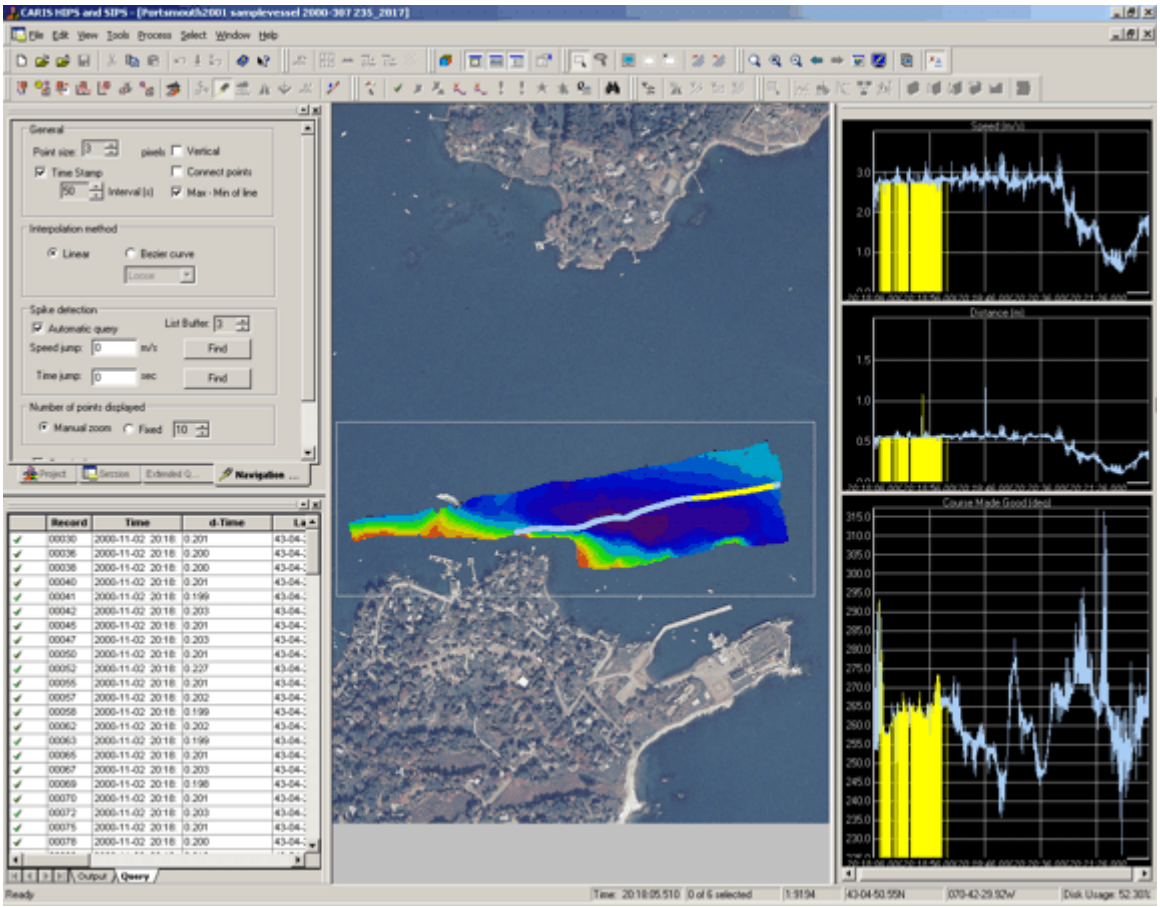
The Navigation Editor lets you examine and clean the position data for both the ship and towfish.

1. Select a track line.
2. Select the Navigation Editor command.

Menu	Tools > Navigation Editor
Tool	

The Navigation Editor is displayed in the interface (see below).

The Navigation Editor shows the track lines as a series of continuous symbols, with each point representing a navigation fix. The CARIS feature code used to represent the navigation symbols is selected in the Options dialog box (see "GENERAL" ON PAGE 75).



The editor contains three time-series graph views of information that is calculated between points in the recorded navigation data:

- **Speed:** Calculated speed.
- **Distance:** Calculated distance.
- **Course Made Good:** Calculated direction (in degrees) from one navigation fix to the next.

The amount of data displayed in the three views is determined by the number of navigation points visible in the Display window.

Resize data points

The point symbols can be resized to a maximum of 10 pixels.

Menu	Window > Control
Tool	

1. Open the Control window.
2. Select the *Navigation* tab.
3. Click the up or down arrow buttons beside the *Point Size* box to change the size of the symbols.

The view in the Display window is refreshed so the navigation data is shown in the new point size.

Time Stamps

Display lines to represent time stamps in the Navigation Editor windows. The time stamp interval is displayed in seconds and can be changed.

1. Select the *Time Stamp* check box to display time stamp lines.
2. Click the up or down arrow buttons beside the *Interval* to select the gap (in seconds) between time stamps.

Set minimum/maximum of line

When this option is selected the vertical scaling of the graphs fits the maximum and minimum range of values for the entire line. Otherwise the vertical scaling of the graphs fits the max-min range of the data currently in view.

1. Open the Control window.
2. Select the *Navigation* tab.
3. Select the *Max-Min of Line* check box.

Number of points displayed

This option determines the number of points visible in the Display window.

- When *Manual* is selected, use the zoom tools to determine how many navigation points are visible in the Display window.
- When *Fixed* is selected, a set number of points are displayed in the window.

1. Open the Control window.
2. Select the *Navigation* tab.
3. Select either the *Manual* or *Fixed* check boxes (Manual is selected by default).
4. If you selected the *Fixed* check box, enter the number of navigation points to be displayed by clicking the up or down arrow buttons.

Connect points

Join the displayed navigation point symbols with a line representing the current interpolation method.

1. Open the Control window.
2. Select the *Navigation* tab.
3. Select the *Connect Points* check box.

The navigation points are connected by line drawn according to the current interpolation method of the line.

Gyro indicator line

A gyro indicator line that shows vessel heading can also be displayed in the Navigation Editor. This feature is useful for detecting dramatic changes in heading that could effect data acquisition.

1. Select the *Gyro Indicators* check box to display the gyro line in the Display window.
2. Click the up or down arrow buttons to select the size (in pixels) of the gyro indicator line.
3. Select a colour for the line from the *Colour* picker, or create a custom colour using the standard Windows Colour Palette.

Line Interpolation

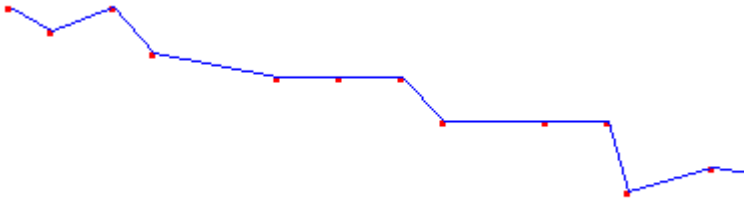
The merge and mosaic functions combine the cleaned bathymetry and imagery data with other sensors and the navigation data. At this stage, the navigation data must be interpolated based on the time of the soundings and side scan data. You can interpolate positions in linear or Bezier fashion.

- **Linear**: Calculate new positions by connecting consecutive positions using straight lines.
- **Bezier**: Calculate new positions by connecting points using a Bezier curve that does not necessarily connect all navigation positions. Bezier curves are available in three types: tight, medium and loose.

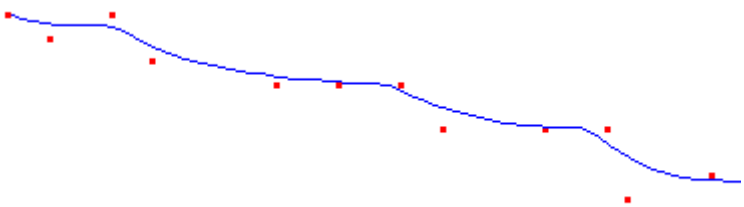
If you do not select an interpolation method then the default is linear. Linear interpolation is suitable if the original navigation positions are clean and do not significantly deviate from neighbouring positions. Bezier interpolation is suitable if the original data is noisy.

Below are examples of the interpolation options.

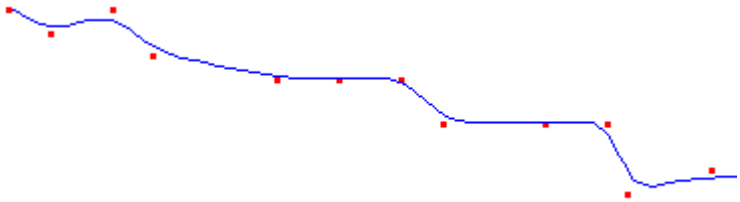
Linear Interpolation.



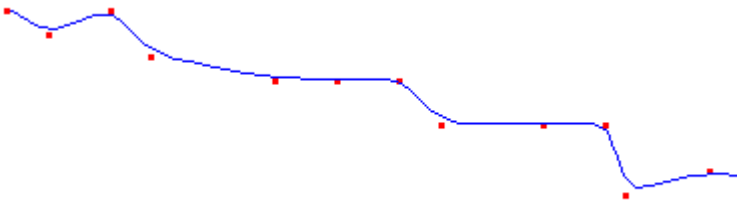
Bezier Interpolation: Loose



Bezier Interpolation: Medium



Bezier Interpolation: Tight



Select a line interpolation option

Menu	Window > Control
Tool	

1. Make sure the Control window is open.
2. Select the *Navigation* tab.
3. Select the *Connect Points* check box.

A line is drawn through the points in the Display window.

4. Select either *Linear* or *Bezier Curve* by selecting the appropriate check box.
5. If the *Bezier Curve* option is enabled, select either a *Loose*, *Medium* or *Tight* format from the drop-down menu.

The line connecting the points is displayed according to the selected options.

The Set Interpolation command lets you select an interpolation option for one or more survey lines without opening the Navigation Editor. See “NAVIGATION” ON PAGE 248.

Examine Navigation Data

Navigation outliers can affect the final positions of the soundings during the Merge process. Therefore, the navigation data should be (but is not required to be) examined and cleaned for any outliers. You can examine the track line to accept, reject or query data, as necessary. There are two methods for locating navigation outliers:

- Examine the track lines and time graphs that show the computed speed, distance, and course made good between navigation points.
- Find speed and time jumps using the search functions in the Control window.

Individual position fixes

1. Click the scroll bar at the bottom of the Navigation Editor windows, or select a Pan command (see "PAN" ON PAGE 39).

The view in the Display window is refreshed to accommodate the location along the track line. The relevant speed, distance, and course-made-good data is displayed in the Navigation Editor time graphs.

2. Select data, if necessary, in either the Display window or the time graphs by pressing and holding the mouse button while dragging the cursor across the data.

The selected data is highlighted.

3. Release the mouse button when finished.
4. Select a reject (see "REJECT DATA" ON PAGE 269), accept (see "ACCEPT DATA" ON PAGE 269) or query (see "QUERY DATA" ON PAGE 270) option.

Speed jumps

The Speed Jump option searches the track line for changes in vessel speed that are same as, or greater than, the Speed Jump value.

Menu	Window > Control
Tool	

1. Ensure that the Control window is open and the Navigation tab is displayed.
2. Select the *Automatic Query* check box if you want to display the selected position fixes in the Query tab of the Worksheet window.
3. Enter a value in the *Speed Jump* check box.
4. Click the *List Buffer* up or down arrow keys to select the number of adjacent position fixes that are highlighted when a jump is found. For example if you choose 5, the two position fixes to the right and left of the selected fix are also selected.
5. Click **Find**.
6. Select Accept ("ACCEPT DATA" ON PAGE 269), Query ("QUERY DATA" ON PAGE 270) or one of the Reject "REJECT DATA" ON PAGE 269) options.
7. Click **Find** again to move along the track line.

Time jumps

The Time Jump option searches the track line for changes in time (in seconds) that are the same as, or greater than, the Time Jump value.

1. Ensure that the Control window is open and the *Navigation* tab is displayed.
2. Select the *Automatic Query* check box if you want to display selected position fixes in the Query tab of the Worksheet window.
3. Enter a value in the *Time Jump* box.
4. Click the *List Buffer* up or down arrow keys to select the number of adjacent position fixes that are highlighted when a jump is found. For example if you choose 5, the two position fixes to the right and left of the selected fix are also selected.
5. Click **Find**.
6. Select one of the Reject options ("REJECT DATA" ON PAGE 269), Query ("QUERY DATA" ON PAGE 270) or Accept ("ACCEPT DATA" ON PAGE 269).

Reject, Accept and Query Data

Reject Data

Select from two reject data options:

- **Reject-With Interpolation:** Soundings associated with this navigation data are processed further since they have positions interpolated for them.
- **Reject-Break Interpolation:** Soundings associated with this navigation data are flagged as rejected and cannot be processed.

1. Select the data so it is highlighted.
2. Select a Reject command.

The selected data is now flagged as rejected.

Menu	Edit > Status Flag > Reject-With Interpolation/Reject-Break Interpolation
Tool	
Pop-up	Reject > Reject-With Interpolation/Reject-Break Interpolation

Accept Data

All data is marked as accepted until flagged otherwise. Use this option to change the status of rejected data.

1. Select the data so it is highlighted.
2. Select the Accept command.

The previously rejected navigation data is now flagged as accepted.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

Query Data

Examine data for further information.

1. Select the data so it is highlighted.
2. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

Information about the position fixes is displayed in the Worksheet window. The fixes can be rejected or accepted in this window.

Query Line

The Query Line command lets you view navigation data for an entire track line.

1. Select a track line.
2. Select the Query Line command.

Menu	Edit > Query Line
Pop-up	Query Line

Navigation points for the selected line are displayed in the Worksheet window. You can change the status of a navigation point by right-clicking the mouse and selecting a reject or accept option.

10

Attitude Editor

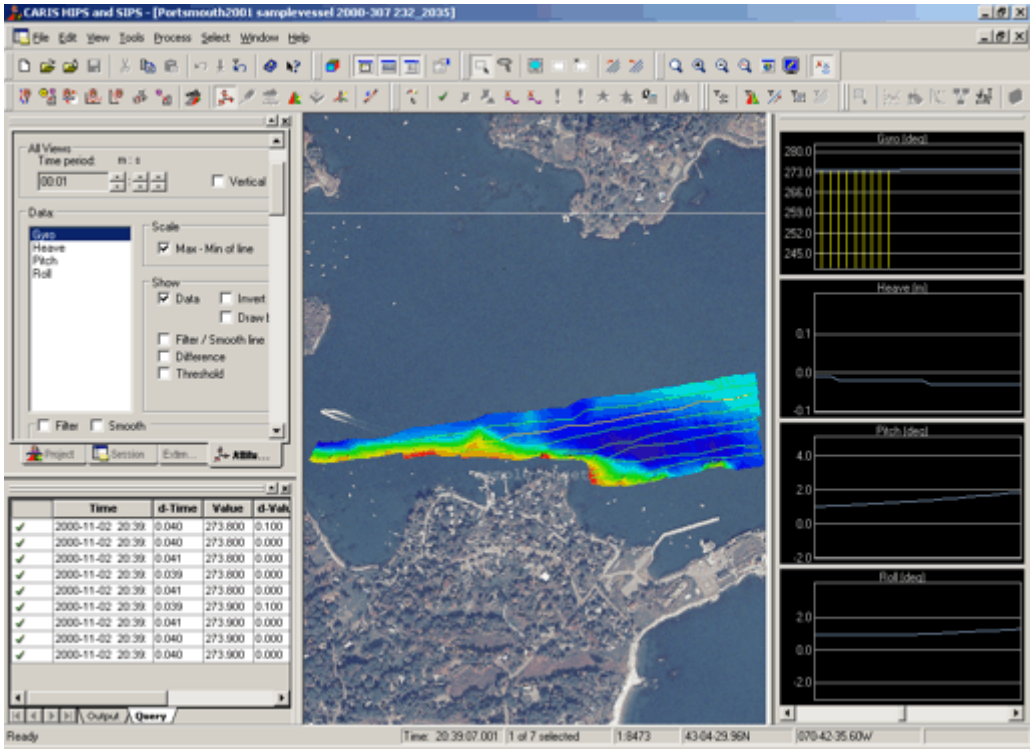
Attitude Editor Interface

The Attitude Editor displays sensor data related to the movement of a vessel or towed sensor. It also displays data from other auxiliary sensors that use a simple time/value data model. The data is plotted in chronological order and is time stamped. You can edit data directly in the editor or set filter parameters to reject unwanted data.

1. Select a track line so it is highlighted.
2. Select the Attitude Editor command.

Menu	Tools > Attitude Editor
Tool	
Pop-up	Attitude Editor

The Attitude Editor is displayed in the interface (as shown below). You can open the Attitude Editor while the Swath Editor is active and examine attitude and sounding data at the same time (see "SWATH EDITOR" ON PAGE 297).



The Project-Vessel-Day directory and file name of the selected track line is displayed on the title bar.

Data is graphed in the windows according to an X-Y axis. In the default layout, the X-axis represents time and the Y-axis represents data.

By default, the following data is displayed in the editor:

- **Gyro:** Gyro observations are displayed in degrees from positive 1 to 360.
- **Heave:** Heave observations are displayed according to the Units setting in the Options dialog box and are positive as the vessel moves upward.
- **Pitch:** Pitch observations are displayed in degrees and are positive when the bow is down.
- **Roll:** Roll observations are displayed in degrees and are positive when the starboard side is up.

However, this is not the only data that can be displayed in the editor. All sensor data in HIPS/SIPS format can be displayed except for depths, navigation, and side scan data.

Besides the above sensors, the following sensors can be displayed (if available):

- loaded tide
- side scan gyro
- side scan cable out
- side scan depth
- side scan height
- delta draft
- GPS height
- GPS tide
- speed on water

Applanix's MS POV systems calculate TrueHeave values from real-time heave values. The computed TrueHeave data and real-time heave values are stored in a binary file that is not converted to HIPS/SIPS format when other data is imported. These files have to be separately loaded to the track line. For more information, see "ATTITUDE SENSOR DATA" ON PAGE 235.

Display Options

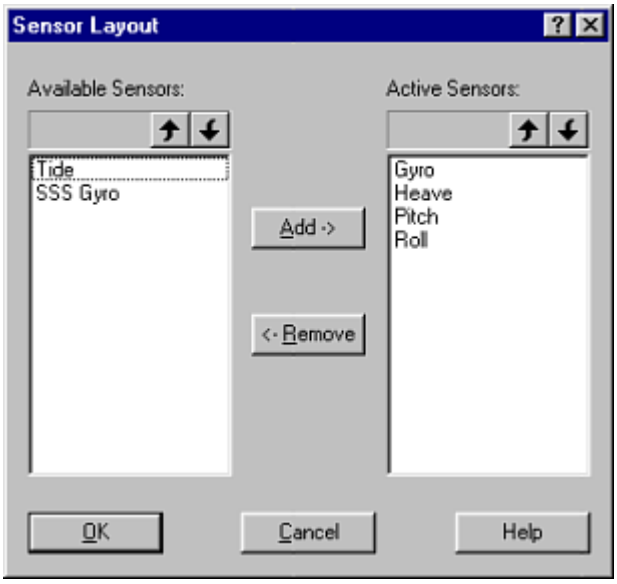
Show/hide sensor data

You can control which attitude sensors are displayed.

1. Select the Sensor Layout command.

Menu	Tools > Sensor Layout

The Sensor Layout dialog box is displayed.



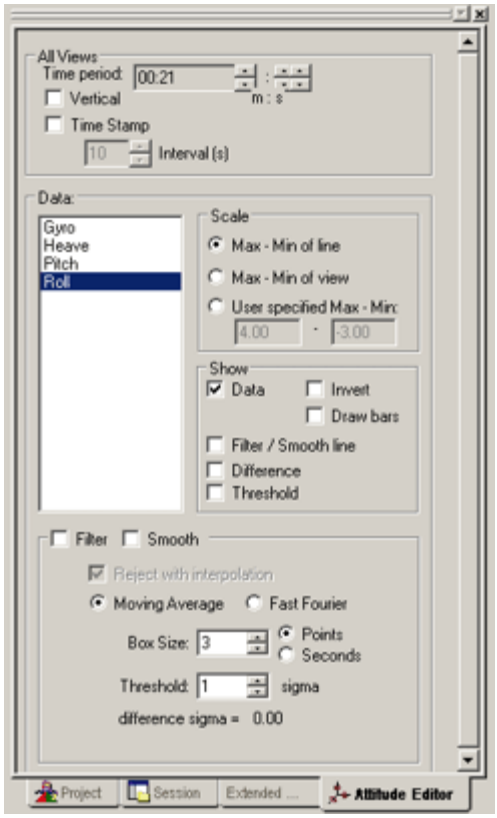
2. To remove a sensor from the *Active Sensors* list, select the sensor and click **Remove**.
3. To add a sensor to the *Active Sensors* list, select a sensor from the *Available Sensors* list and click **Add**.
4. Set the order that sensor windows are displayed by selecting a sensor and clicking the up or down arrow buttons, or by selecting the sensor and dragging it to a new position.
5. Click **OK** to apply settings.

The Attitude Editor is displayed according to the selected settings.

The display of data in the Attitude Editor windows is controlled through the Attitude Editor tab in the Control window. This tab is visible when the Attitude Editor is opened.

1. Open the Control window.
2. Select the *Attitude Data* tab if the tab is not already displayed.

Menu	Window > Control
Tool	



Time

By default, time is displayed on the X-axis. As the time scale is increased, more data is displayed in the graphs; as the time is decreased, less data is displayed.

3. Click the up or down arrows beside the *Time Period* box to set time in minutes and/or seconds.
4. Select the *Time Stamp* check box to display the time stamp in the windows.
5. Click the *Time Interval* up or down arrow buttons to select an interval for time stamp display.

The display in all graphs is adjusted to the new time setting.

Scale Data

By default, the data scale is shown on the Y-axis. The scale of the data axis can be based on the minimum and maximum values of data in the entire line, the minimum and maximum values of the data currently in view, or by a user-defined axis.

1. Select a sensor from the *Data* list.
2. Select from one of three scaling options:
   - Select *Max-Min of Line* to scale data according to the minimum and maximum data values of the entire line.
   - Select *Max-Min of View* to scale data according the minimum and maximum values currently in view.
   - Select *User-defined Max-Min* to define maximum and minimum scale and type the values in the maximum and minimum fields.

Vertical view

The *Vertical* option switches the data graph windows from a horizontal layout to a vertical layout. In this mode, the X-axis represents the data values and the Y-axis represents time.

1. To view the Attitude Editor in vertical mode, click the *Vertical* check box.

The Attitude Editor windows are displayed in vertical mode.

Other Display Options

Invert view

The *Invert* command displays the data in a “mirror-image” view. This function is useful for viewing towfish depth in the water column.

1. Select an attitude data type in the *Data* list.
2. Select the *Invert* command.

The data is shown as inverted in the appropriate section.

Draw bars

The *Draw Bars* function displays vertical bars for each sensor value in the graphs. This option makes it easier to see time gaps between sensor values.

1. Select the *Draw Bars* check box in the Attitude Data tab of the Control window.

The *Smooth/Filter Line*, *Difference* and *Threshold* options are described in the Shows Graphs section of "SMOOTH DATA" ON PAGE 278

Colour Display

The colour for displaying the various graph lines can be changed.

Menu	Tools > Options

1. Select the Options command.

The Options dialog box is displayed.

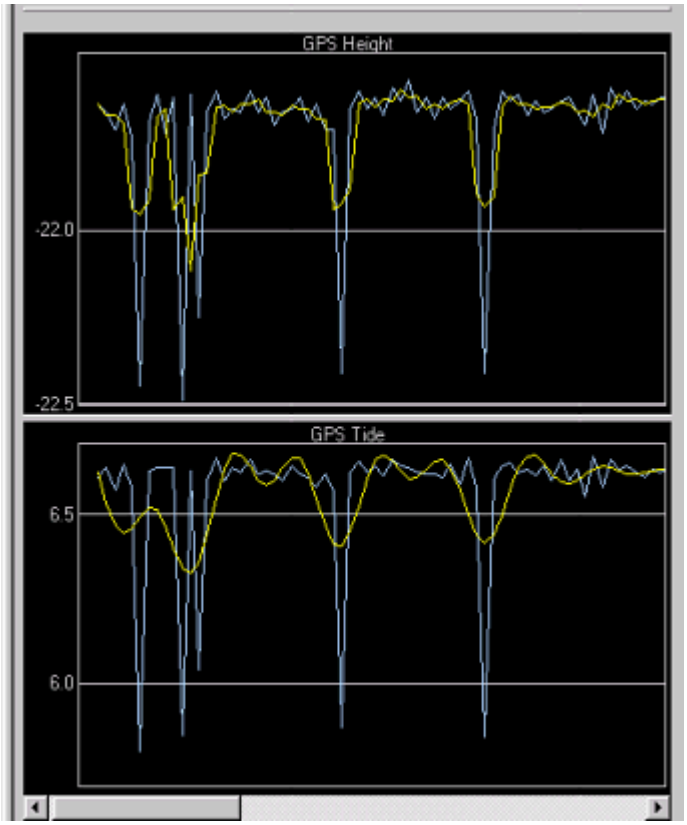
2. Click the Display tab and click the *Sensor Editors* category so it is highlighted.
3. Select a colour for the sensor filter line from the drop-down colour palette, or create a custom colour from the standard Windows colour picker.
4. Click **OK**.

See "DISPLAY" ON PAGE 78 for more information on colour options for the Attitude Editor.

Smooth Data

The Smooth Sensor function is used to smooth out localized variability. Any sensor data that can be viewed in the Attitude Editor can be smoothed.

Below is an example of GPS height and tide.



The blue line represents the original data and the yellow line represents the smoothed data. The parameters for creating the smoothed data are saved to the SmoothedCoefficients file in the HCDS_Data\Project\Vessel\Day\Line folder. The file is in XML format. This file is applied to the track line during the merge process and SVP Correction.

Moving Average and Fast Fourier Transform

There are two methods available for smoothing: Moving Average and Fast Fourier Transform (or FFT).

- The *Moving Average* function calculates an average sensor value over a window of data using the Box Size parameter. The window is defined as a number of data points or seconds centred on a point. The average value for all of the values in the window is calculated and will be given to the central point.

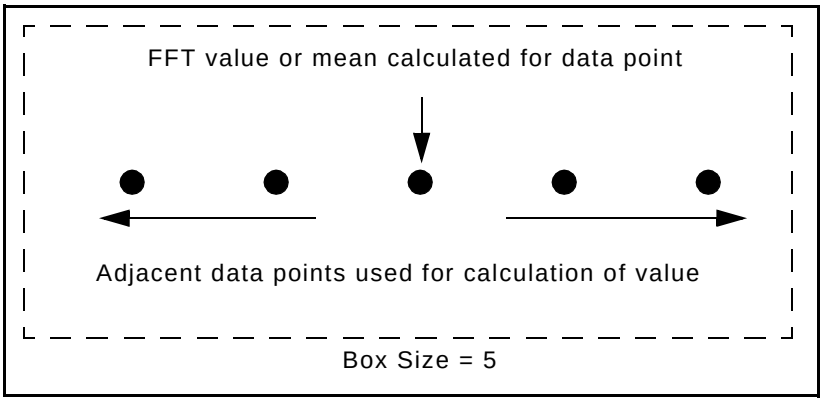
- The *Fast Fourier Transformation* performs a low-pass filter on the selected sensor. You select a Box Size in either points or seconds. This value is converted into seconds and then inverted to become the cut-off frequency. Increasing the Box Size results in greater smoothing.

Box size

The *Box Size* (Step 6) determines how many neighbouring data points are used in the smoothing calculation. The number of neighbouring data points can determined using time or a set number of points.

Note the different ways the Moving Average and the FFT use the Box Size parameter. The Moving Average function uses the Box Size to determine the window that is used to calculate the average for the new “smoothed” centre point. The FFT algorithm uses the Box Size to define the cut-off frequency that is then applied to all data.

In the following example, the box size is set to five data points. A FFT value or mean for any data point is calculated from itself and ‘the two data points on either side of it (if available).



1. Make sure that the Control window is open and the Attitude Data tab is visible.
2. Select a sensor in the *Data* list box, or click inside a sensor window in the Attitude Editor.
3. Select the *Smooth* option.
4. Select either the *Moving Average* or *Fast Fourier* smoothing options.
5. Select either the *Points* or *Seconds* check boxes.
6. Click the *Box Size* up or down arrow buttons to select the number of adjacent data points or a time range.

Show graphs

7. Show any of the four graphs by selecting the appropriate check boxes:
   - *Data* graphs the raw sensor data. This box is checked by default.
   - *Filter/Smooth Line* graphs the computed values derived from the adjacent data points as described above.
   - *Difference* graphs the difference between the original data values and the moving average or FFT value for each data point.
   - *Thresholds* display horizontal lines in the graph indicating the difference values that are clipped during smoothing. Threshold is a multiple of the standard deviation (sigma).

     The upper threshold signal is determined by the following formula:

     ```
     original mean value + (threshold value X the
     standard deviation)
     ```

     The lower threshold signal is determined by the following formula:

     ```
     original mean value - (threshold value X the
     standard deviation
     ```

The smoothing function applies the results only to the Data graph and Filter/Smooth Line graph.

The line is smoothed according to the parameters set above. When you close the Attitude Editor, you are prompted to save any changes. This saves any smoothing changes to the SmoothedCoefficients file.

8. Click **Yes** to save smoothed data.

Cleaning Data

You can examine the attitude data to determine if it should be rejected, accepted, or queried for further information.

Reject data

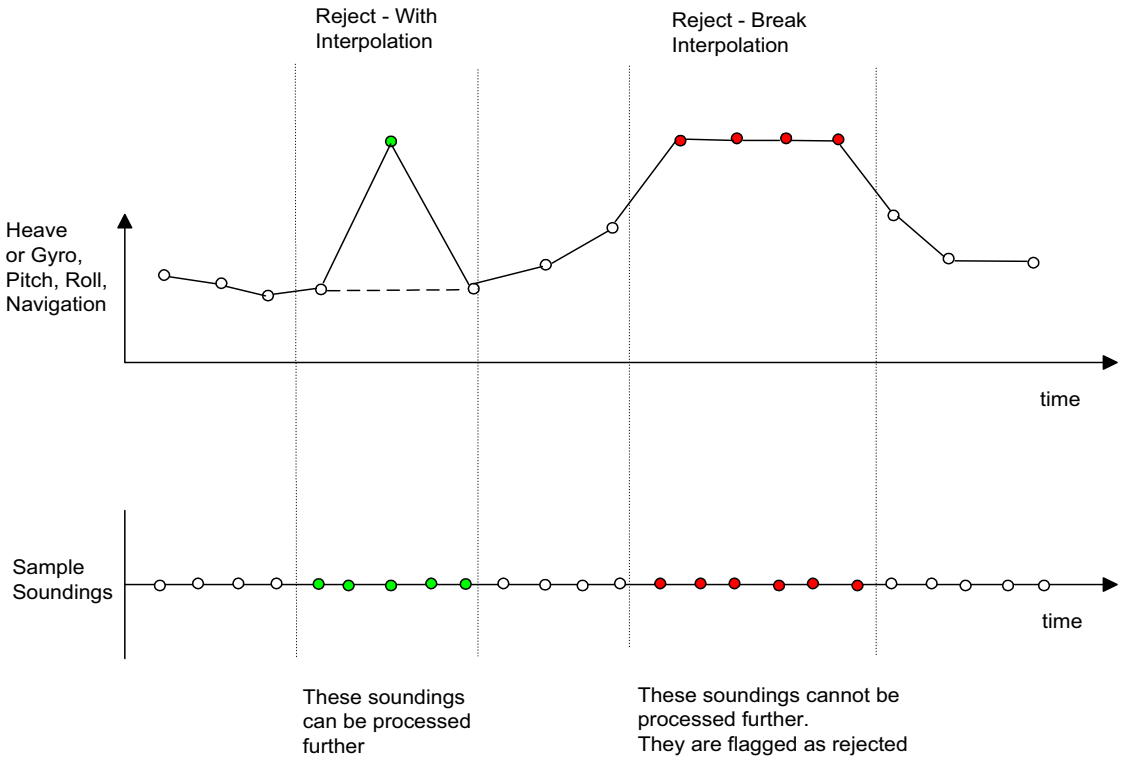
1. In any of the Attitude Editor windows, press and hold the mouse button while dragging the cursor across the data to be examined.

The selected data is highlighted.

2. Select one of the following Reject options:
   - **Reject- Break Interpolation:** When the files are merged, the Break Interpolation option rejects all soundings recorded at the same time as the rejected sensor data.
   - **Reject - With Interpolation:** When the files are merged, the With Interpolation option rejects the highlighted sensor data but uses neighbouring data points for interpolation so sounding data is not rejected.

Menu	Edit > Reject-Break Interpolation/Reject-With Interpolation
Tool	
Pop-up	Reject-Break Interpolation/Reject-With Interpolation

Below is a graphical representation of the reject options and sounding data.


Reject - With Interpolation
Reject - Break Interpolation
Heave or Gyro, Pitch, Roll, Navigation
time
Sample Soundings
time
These soundings can be processed further
These soundings cannot be processed further. They are flagged as rejected

For some multibeam sonars, the data converted into HIPS has already been compensated for heave, pitch, and roll. Therefore heave, pitch, and roll are not applied during Merge as indicated by the “Apply” options in the vessel configuration. However, it is still possible to use Reject–Break Interpolation on the recorded heave, pitch, and roll data which results in the sounding data during the same time period being rejected by Merge.

You can show or hide rejected data. When the rejected data is turned off, these soundings are no longer visible in the Attitude Editor.

1. Select the Show Rejected command.

Menu	View > Show Rejected
Tool	
Key	<F4>

At any time, attitude data that has been flagged as rejected can be converted to accepted again by selecting the Restart Cleaning command (see “RESTART CLEANING” ON PAGE 266).

Accept Data

All data is considered as accepted until explicitly rejected. However, sometimes it is necessary to change previously rejected soundings to accepted again.

1. Click and hold the left mouse button while dragging the cursor across the data to be selected.

The data is highlighted.

2. Select the Accept command.

Menu	Edit > Accept
Tool	
Pop-up	Accept
Key	<A>

Previously rejected sensor data is now flagged as accepted.

Query data

Querying displays information about selected data.

1. Press and hold the left mouse button while dragging the cursor across the data to be queried.

The data is selected.

2. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

The following data is displayed in the Worksheet window:

- time stamp
- d-time difference in seconds from last time stamp
- sensor value in either degrees or metres/feet
- d-value difference from last value
- status

The following two data values are related to the smoothing and filtering options (see "SMOOTH DATA" ON PAGE 278 and "FILTER SENSOR DATA" ON PAGE 284):

- averages derived from adjacent data points
- difference between original data values and moving average or FFT value for each data point

Change data status flag to rejected or accepted.

1. Select the data in the Worksheet window so it is highlighted.
2. Select a Reject or Accept command.

The data is marked as accepted or rejected, depending to the option you selected.

Query line

Display all attitude sensor data for a line.

1. Select one of the attitude sensors.
2. Select the Query Line command.

Menu	View > Query line
Pop-up	Query Line

Information about the selected sensor is displayed in the Worksheet window. You select data and right-click the mouse to choose an accept or reject command.

Filter Sensor Data

The Attitude Editor filters reject data with residual values that fall outside user-defined threshold limits.

1. Make sure that the Control window is open and the Attitude Editor tab is visible.
2. Select a sensor in the *Data* list, or click inside a sensor window in the Attitude Editor.
3. Select the *Filter* check box.

By default, data is rejected without interpolation so that soundings with the same time stamp as the rejected data are also flagged as rejected. You can choose to reject data with interpolation

4. Clear the *Reject with Interpolation* check box so that soundings associated with the rejected sensor data are also rejected.

There are two filtering options: *Moving Average* and *Fast Fourier*. The *Moving Average* calculates a mean for each data point and *Fast Fourier* applies a common wave-smoothing algorithm to a data point.

5. Select either the *Moving Average* or *Fast Fourier* filtering option.

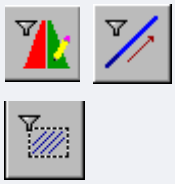
The *Box Size*, *Time* and *Point* options work in exactly the same manner as in the Smoothing function. See "SMOOTH DATA" ON PAGE 278 for more information.

6. To select how the average is calculated, select either the *Points* or *Seconds* check boxes.
7. Click the *Box Size* up or down arrow buttons to select the number of adjacent data points or to set a time range.
8. Show any of the four filter graphs by selecting the appropriate check boxes (see "SMOOTH DATA" ON PAGE 278)
9. Click the *Threshold* up or down arrow keys until you reach a desired value.

The Thresholds are indicated by lines positioned above and below the Difference graph.

Reject data

Reject all Difference data that falls outside the Thresholds by selecting a filter option.

Menu	Tools > Apply Filters > 1 Screen/To End of Line/Selected Lines
Tool	

10. Select a Filter command.

The corresponding attitude data is rejected.

11

Sound Velocity Editor

Overview

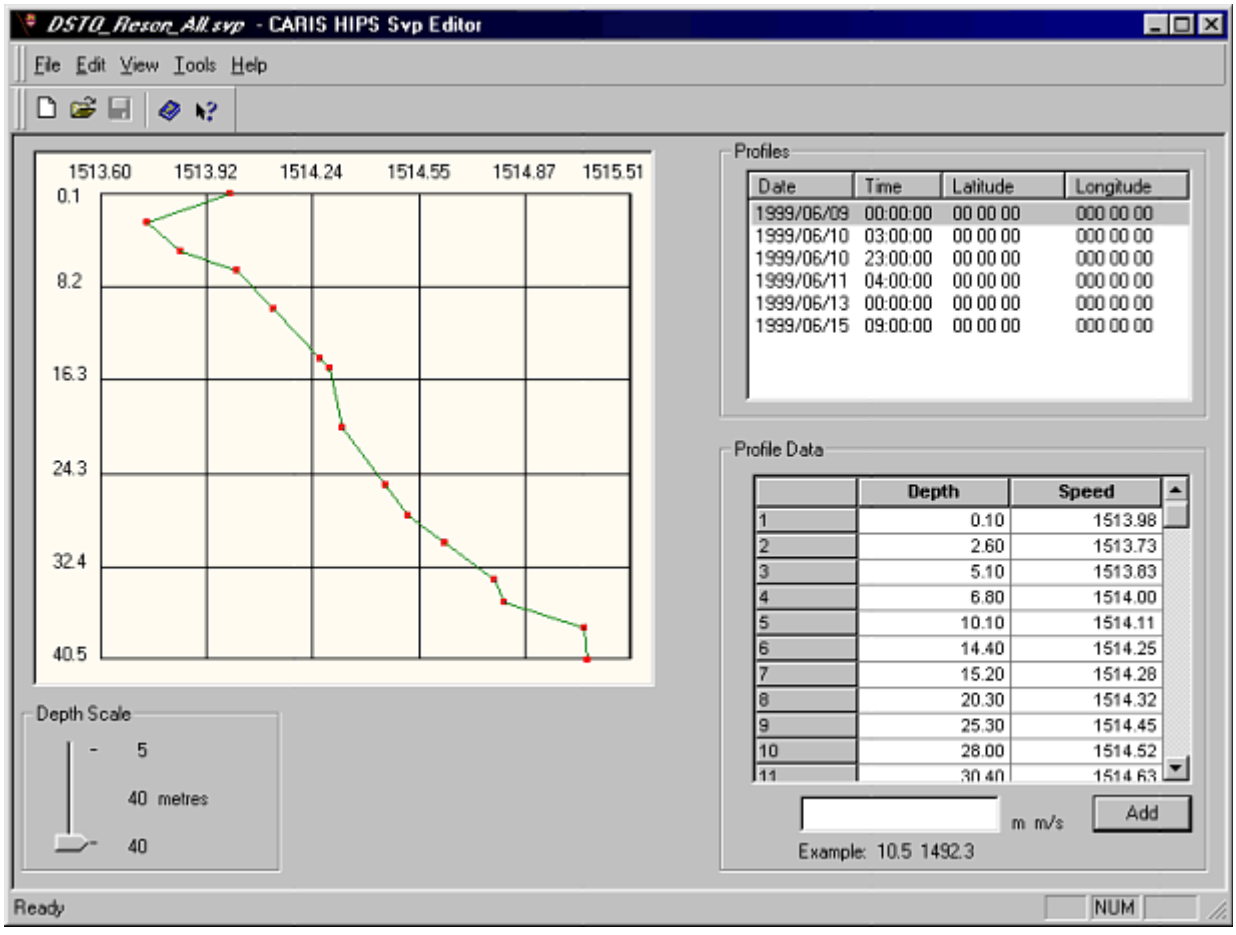
Sound velocity processing in HIPS is available for sonar formats that provide raw uncorrected travel time and angles from the transducer. Multibeam formats that meet these requirements include the XTF, XSE and Hypack formats. Also, single beam data converted using the HIPS Generic Data Parser can be corrected for sound velocity.

The SVP Editor helps you to edit existing SVP data, or create new data. The editor is a separate program that is launched from HIPS and SIPS.

Menu	Edit > Sound Velocity Profiles
Tool	

1. Select the SVP Profiles command.

The SVP Editor is displayed.



The editor contains a section with a list of profiles. The profiles include a latitude and longitude position for applying the *Nearest*

in Distance option. Below the *Profiles* section is a table of the depth/speed values for a selected profile. The left-hand side of the editor contains a graphical representation of the depth/speed data.

Open an Existing File

Menu	File > Open
Tool	
Key	<Ctrl+O>

1. Select the Open File command.

The Open dialog box is displayed. All SVP files are located, by default, in ..\Hips\Svp.

2. Select an SVP file so it is highlighted, or type the file path in the *File Name* text box.
3. Click **OK**.

A list of profiles is displayed. Data in the first profile is displayed in the Profile Data table and the speed/depth graphed area.

4. To navigate between profiles, click the time stamps in the *Profiles* list box.

Depth Scale

The *Depth Scale* slider determines the view for the depth/speed graphed area by setting a maximum depth value. You can change the vertical display of data by adjusting the depth scale.

1. Position the cursor over the *Depth Scale* slider and press and hold the mouse button, while dragging the cursor up or down so the slider moves with the cursor.

As the slider is moved, the view of the depth/speed data adjusts to the new depth scale.

Create a New SVP File

You can create a new SVP file.

Menu	File > New
Tool	
Key	<Ctrl+N>

1. Select the New File command.
2. Create new SVP profiles (see "CREATE NEW PROFILE" ON PAGE 295).
3. Create new depth/speed values, as needed (see "ENTER NEW DEPTH/SPEED DATA" ON PAGE 293).

Menu	File > Save As

4. Select the Save As command.
5. Type a name for the SVP file in the file name box.
6. Click **Save**.

The file is saved the directory you selected.

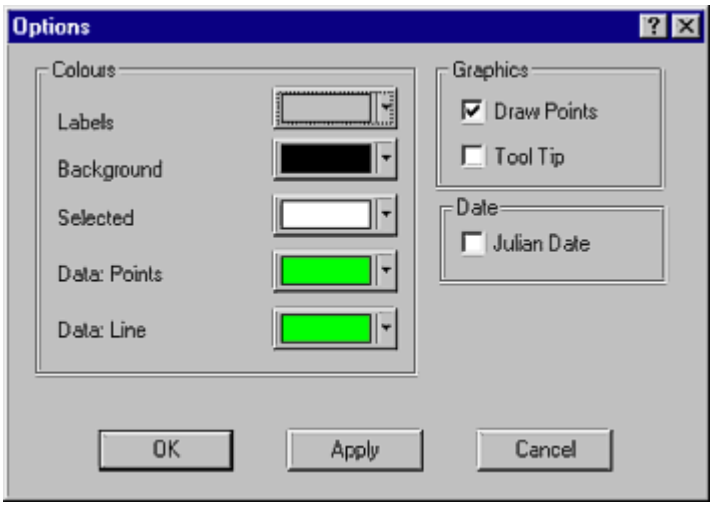
Options

You can select the colours used for the graphical display.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.



Colour Options

2. Select a colour from the drop-down palette; or create a custom colour from the standard Windows colour picker.
3. Click **OK**.

The display is refreshed to show the selected colours.

Draw points

Show or hide the depth/speed data points in the graphical display.

4. Select the *Draw Points* check box.

If the box is checked, the data points are visible in the graphical display.

5. Click **Apply** to apply and save changes to the current file.
6. Click **OK**.

Show/hide tool tips

A tool tip is a label containing the depth/time position for a data point in the graphical display. The label becomes visible when the mouse cursor is moved in the graphical display.

7. Select the *Tool Tips* check box to show or hide the tool tips.

If the box is checked, the depth/time labels are visible.

8. Click **Apply** to apply and save changes to the current file.
9. Click **OK**.

Show Julian date

Switch the date fields in the SVP editor between the regular calendar format and Julian calendar format.

10. Make sure the *Julian Date* box is checked if you want to select this feature.

11. Click **Apply** to apply and save changes to the current file.
12. Click **OK** to close the dialog box.

Show/hide toolbar

Display or hide the toolbar.

1. Select the Toolbar command.

Menu	View > Toolbar

If the menu item is checked then the toolbar is visible; if it is not checked, the toolbar is not visible.

Show/hide status bar

Display or hide the status bar.

1. Select the Status Bar command.

Menu	View > Status Bar

If the menu item is checked, the status bar is visible; if it is not checked, the status bar is hidden.

Edit SVP Data

You can use the SVP Editor to modify existing speed/depth values, append new data to an existing file, delete data or delete a complete profile.

Change depth/speed values

Individual values in the depth/speed table can be changed.

1. Position the cursor over any *Depth* or *Speed* cell in the table.
2. Click inside the cell.
3. Type a new value.
4. Click outside the cell.

The table is changed to reflect the new data.

Enter new depth/speed data

Append new data to an existing profile.

1. Click a profile to display the data in the editor.

You must enter all new depth/speed data in pairs. There are two options for entering data:

- separate the depth/speed data with a space: `15 1200`
- separate the data with a comma: `15, 1200`

1. Type new depth *and* speed values in the text box beside the **Add** button.
2. Click **Add**.
3. Repeat Steps 1 through 3 until all necessary values are entered.

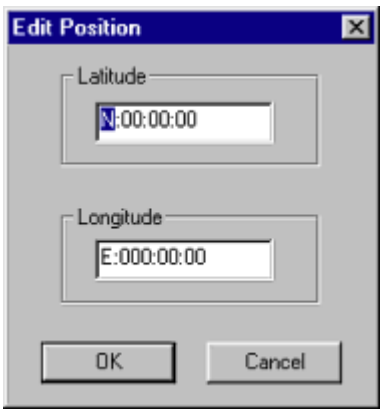
Edit geographic coordinates

Enter new latitude and longitude positions for SVP data. This information is important if you want to use the *Nearest in Distance* option for sound velocity correction (see "SOUND VELOCITY PROCESSING" ON PAGE 241).

1. Click a time stamp beside a profile.
2. Select the Edit Position command.

Menu	Edit > Edit Position

The Edit Position dialog box is displayed.



3. Click the latitude coordinates and type new ones.

4. Click the longitude coordinates and type new ones.
5. Click **OK**.

The new geographic coordinates are displayed in the *Latitude* and *Longitude* sections of the Profiles area.

Delete speed/depth data

1. Press and hold the mouse button, while dragging the cursor over the data points in the graphical display; or click a row number in the table.

The selected data is highlighted.

2. Select the Delete command.

Menu	Edit > Delete
Key	<Delete>

The Speed/Depth data points are removed from the graphical display and the values are no longer displayed in the table.

Remove Profiles

Remove a currently open profile from the SVP editor.

1. Select a profile by clicking the time stamp in the *Profiles* list box.

The depth/speed values are displayed in the editor.

2. Select the Remove Profile command.

Menu	Edit > Remove Profile
Key	<Ctrl+Delete>

The profile is removed from the editor.

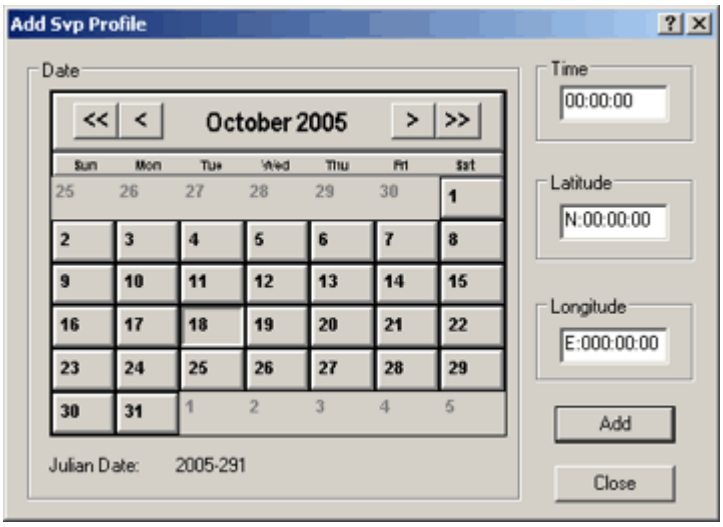
Create New Profile

New SVP profiles can be created through the SVP Editor.

1. Select the Add Profile command.

Menu	Edit > Add Profile
Key	<Ctrl+Insert>

The Add SVP Profile dialog box is displayed.



2. Click the arrow buttons at the top of the calendar to select a month and year (all profiles must be time stamped).
3. Click a day button on the calendar to select a day for the file.
4. Enter the new hour/minute values (use the 24-hour clock) by clicking each section in the *Time* box and typing new values.
5. Enter latitude and longitude coordinates in the appropriate boxes, if needed.
6. Click **Add** to apply settings.

The new file information is displayed in the *Profiles* box.

7. Enter new speed/depth data, as needed (see "ENTER NEW DEPTH/ SPEED DATA" ON PAGE 293).

Save a file

When you close the editor, you are prompted to save the changes.

8. Select a Save command.

Menu	File > Save/Save As
Tool	
Key	<Ctrl+S>

Exit SVP editor

Close the SVP Editor.

Menu	File > Exit

1. Select the Exit command.
2. If you have not saved any changes you are prompted to do so.

The SVP Editor is closed.

12

Swath Editor

Overview

The Swath Editor provides data-cleaning tools for multibeam data, multi-transducer data and Lidar data (see "LIDAR DATA" ON PAGE 469).

The Swath Editor is a graphical tool containing multiple views of the data with many controls for scaling, exaggeration, colours, and so on.

Data cleaning consists of interactively selecting and rejecting soundings and using proven and efficient filtering functions to automatically detect and reject outliers.

You can also use the Swath Editor to flag Designated Soundings ("DESIGNATED SOUNDINGS" ON PAGE 223).

Swath Editor Interface

Menu	Tools > Swath Editor > Open
Tool	
Pop-up	Tools > Swath Editor

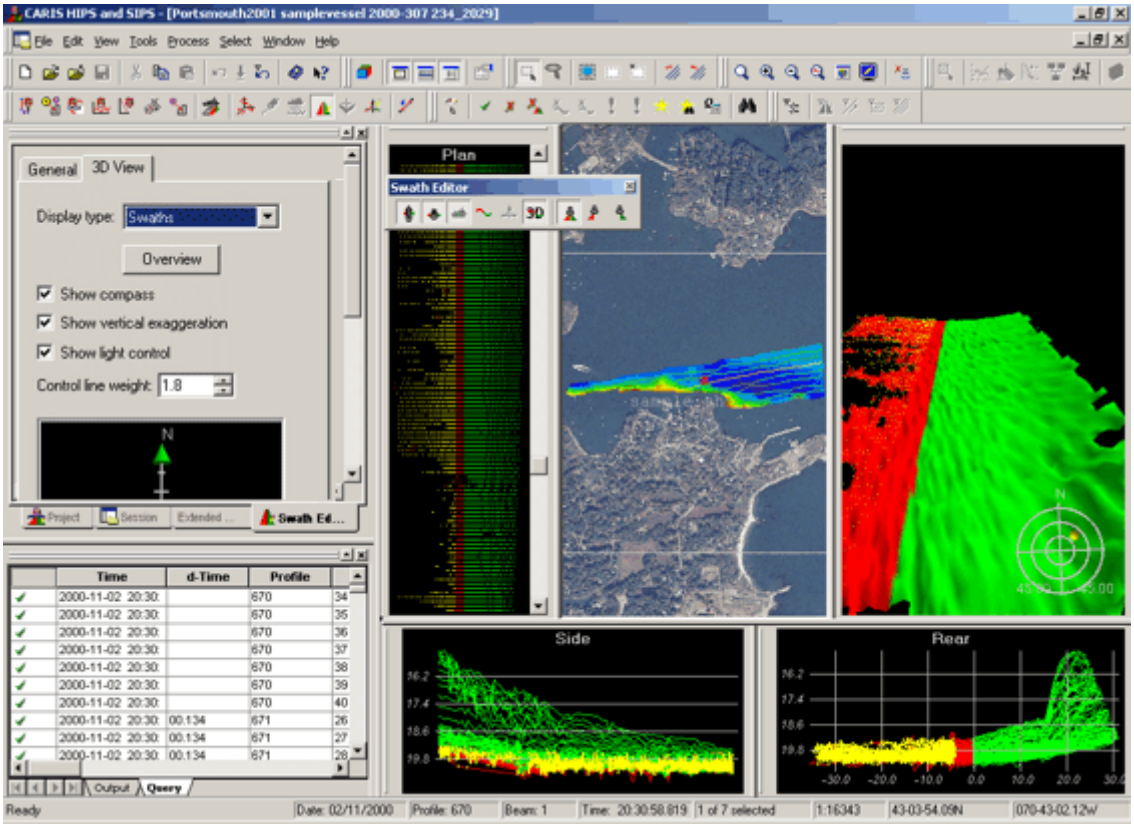
A track line must be selected before the Swath Editor editor can be opened (see "SELECT SURVEY LINES" ON PAGE 219).

1. Select the Open Swath Editor command.

The Swath Editor opens with all views displayed.

The Project-Vessel-Day directory and file name of the selected track line is displayed on the title bar.

HIPS is shown below with the Swath Editor open.



The Swath Editor consists of six views:

- Plan View
- Side View
- Rear View
- Profile View
- 3-D View
- Amplitude View

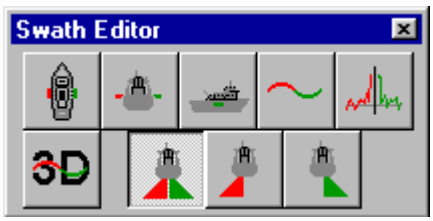
Views can be displayed, hidden or positioned anywhere on the interface. They can also be adjusted to any size you want. (See "REPOSITION AND RESIZE WINDOWS" ON PAGE 22). The layout of the views is remembered by the application when you exit HIPS.

If the dataset contains side scan data, you can also open the Side Scan Editor while the Swath Editor is open (for more information on the Side Scan Editor, see "SIDE SCAN EDITOR" ON PAGE 494).

The tabbed pages in the Control window set the various Swath Editor features.

Toolbars

The display of the views is controlled from the Swath Editor toolbar.



This is a floating toolbar that is automatically displayed when the editor is opened. The toolbar can be docked or floated over any area of the interface. Its appearance is controlled from the Toolbar dialog box (see "MODIFY EXISTING TOOLBARS" ON PAGE 25).

Menu	Tools > Swath Editor > Close
Tool	

1. Select the Close Swath Editor command.

All Swath Editor views close and you are prompted to save any changes.

2. Click **OK** to save changes.

Plan View

The Plan View presents an overhead view of the swath profiles and beams. All profiles are spaced equally in the along-track direction. Across-track beam spacing is to correct scale. Each beam is represented by a square.

1. Select the Plan View command.

Menu	Tools >Swath Editor >Plan
Tool	

The number of swath profiles displayed in the Plan View is determined by three factors:

- height of the view
- point size of the beams
- distance (in points) between swath profiles (see "Data Display in Views" on page 303)

You can apply a number of colour options:

- colour by port and starboard beams (see "Swath Editor" on page 93)
- colour by sonar detection method (not supported by all sonars) (see "Swath Editor" on page 93)
- colour by depth interval
- sun illumination (see "Examine Swath Data" on page 307).

3-D View

The three-dimensional (3-D) View shows all soundings currently visible in the Plan View in three-dimensional format. The data in this window can be rotated and moved around (see "Examine 3-D Data" on page 312).

1. Select the 3-D View command.

Menu	Tools > Swath Editor > 3D View
Tool	

Side View

The Side View presents the survey as seen from the side of the vessel with the vessel travelling from left to right. Each line represents soundings from the same beam.

1. Select the Side View command.

Menu	Tools > Swath Editor > Side
Tool	

You can change the display (see "DATA DISPLAY IN VIEWS" ON PAGE 303) and depth label settings for this view (see "SOUNDINGS" ON PAGE 306).

Rear View

The Rear View presents the survey as seen from behind the vessel. Each line represents one swath.

1. Select the Rear View command.

Menu	Tools > Swath Editor > Rear
Tool	

Profile View

The Profile View displays a single swath.

1. Select the Profile View command.

Menu	Tools > Swath Editor > Profile
Tool	

The time stamp and profile number of the swath are shown in the status bar.

Amplitude View

The Amplitude View displays the beam-averaged amplitude data that is recorded with the swath bathymetry data.

1. Select the Amplitude View command.

Menu	Tools > Swath Editor > Amplitude
Tool	

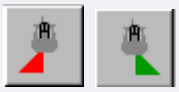
Beam-averaged amplitude data are intensity values for each sounding depth created by a multibeam sonar system.

Data Display in Views

You can show or hide the display of data in the Rear, Side and Profile Views:

- show or hide port beams
- show or hide starboard beams
- show both port and starboard beams (default setting)

1. Choose the Port or Starboard Beams command.

Menu	Tools > Swath Editor > Port Beams/ Starboard Beams
Tool	

The display is refreshed to show only the data that is visible.

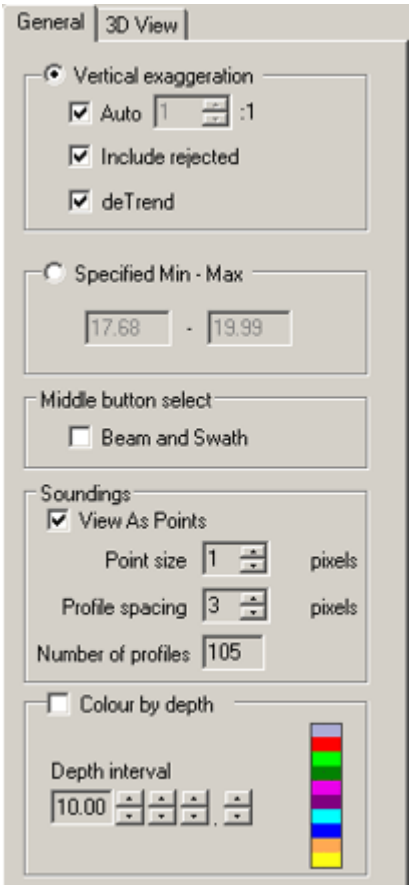
2. To view both starboard and port beam data, select the Port and Starboard Beams command.

Menu	Tools > Swath Editor > Port and Starboard Beams
Tool	

Both Port and Starboard beams are displayed in the Rear, Side and Profile Views.

Swath Editor Display Options

The display options for the views are controlled through the General tab in the Control window.



The following sections describe the options in this tab. The *Middle button select* option is described in "EXAMINE SWATH DATA" ON PAGE 307.

Vertical Exaggeration

The vertical exaggeration is the ratio of the vertical scale versus the horizontal scale used in the Rear and Side Views.

The vertical exaggeration can be determined automatically by including all visible depths or it can be manually set.

When *Auto* mode is on, the *Include Rejected* option controls whether or not the rejected data is used to determine the minimum and maximum sounding depth in the Rear and Side Views.

1. Deselect the *Auto* mode by clearing the check box.
2. Click the up or down arrow buttons beside the *Vertical Exaggeration* box to set the vertical exaggeration.

You can set the vertical and horizontal scale of the view to control the amount of rejected data in the view

3. Select the *Include Rejected* option to resize the display in the view to include rejected data.

This deTrend option is used when the general slopes of the sea floor topography makes it difficult to see individual depth spikes. DeTrend removes the general slope of the sea floor from the Rear and Side views and redraws the swath and beam profiles making it easier to locate depth spikes.

4. Select the *deTrend* check box to view data; select the command by selecting from Tools > Swath Editor > deTrend.

Specified Min-Max

Use a constant scale as an alternative to vertical exaggeration by setting a minimum and maximum range for displaying data.

1. Select the *Specified Min-Max* check box to implement this option.
2. Type the minimum and maximum ranges for the scale.

The views are automatically refreshed to show data according to the scale.

Soundings

By default, beams in the Side, Rear, and Profile Views are connected by lines. Sometimes, beams can be more clearly distinguished if displayed as points.

1. Select the General tab in the Control window.
2. Select the *View as Points* option to change the lines to points.
3. Click the up or down buttons beside the *Point Size* box to set a size value between one and five points.

Increase or decrease the beam display size in the Plan View.

4. Click the up or down buttons beside the *Point Size* box to set a size value between one and five points.

Beams are resized as determined by the point size setting. The *Number of Profiles* box displays a value for the maximum amount of profiles that can be shown in the Plan View. This value is incremented up or down as the beam display size values are changed.

Colour By Depth

Display soundings as colour coded by depth interval.

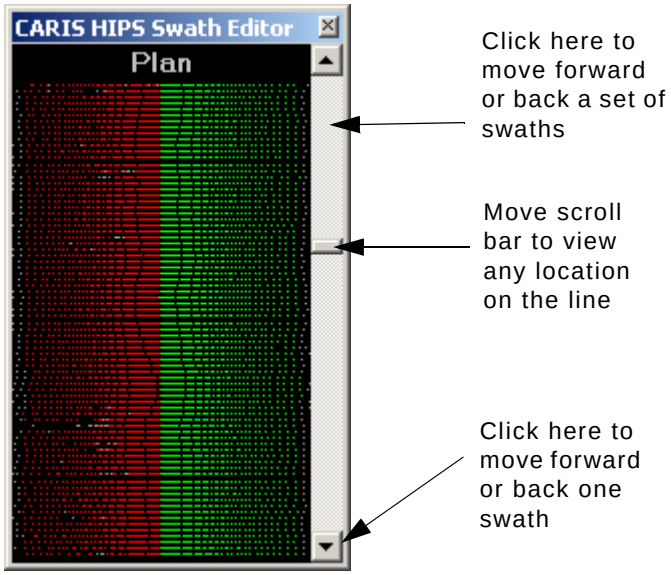
1. Select the Plan View tab in the Control window.
2. Select the *Colour by Depth* box so it is checked.
3. Click the up or down buttons beside the *Depth Interval* box to select a depth interval for the colour band.

The soundings are now displayed by depth level.

Examine Swath Data

The scroll bar in the Plan View represents the time period of the entire line. Each click on an arrow button moves the screen forward or backward one swath. Clicking the grey area of the scroll bar moves the screen forward or backward for a set of swaths. As you move along the track line, all the views refresh to display the same data as in the Plan View window.

Dragging the scroll bar brings in swaths from any location (see example below).


CARIS HIPS Swath Editor
Plan
Click here to move forward or back a set of swaths
Move scroll bar to view any location on the line
Click here to move forward or back one swath

As you move up or down the track line, all the views refresh to display the data from the swaths that are visible in the Plan view.

The geographic location of the swaths currently in view is shown by the sounding coverage outline in the Display window.

As an alternative, use the following methods to scroll through a track line:

- Press the space bar to move up the track line.
- Use the mouse wheel (if available) to move up or down the track line.
- Press <Ctrl + spacebar> to move down the track line.
- Use the mouse wheel (if available) to move up or down a track line.
- Press the <Home> or <End> buttons to move to the start or end of the track line.

Using the middle mouse button

You can select individual swath or beam profiles and see their location in the Display window by moving the cursor in the Plan,

Rear, 3-D or Side Views while holding down the middle mouse button.

1. Make sure the General tab in the Control window is displayed.
2. Place the cursor in either the Plan View, Rear View or Side View and click the middle mouse button.

The selected swath is highlighted.

3. To view both beam and swath profiles, select the *Beam* and *Swath* check box in the *Middle Mouse Select* section.

Individual swaths and beams are highlighted as you move the cursor across the view. The other views are updated to reflect the same beam and swath location. The positions of the swaths and beams are also displayed in the sounding coverage outline.

If the Attitude Editor is open, you can view the sounding's position in relation to attitude data by the yellow line in any of the sensor windows (see "ATTITUDE EDITOR" ON PAGE 271).

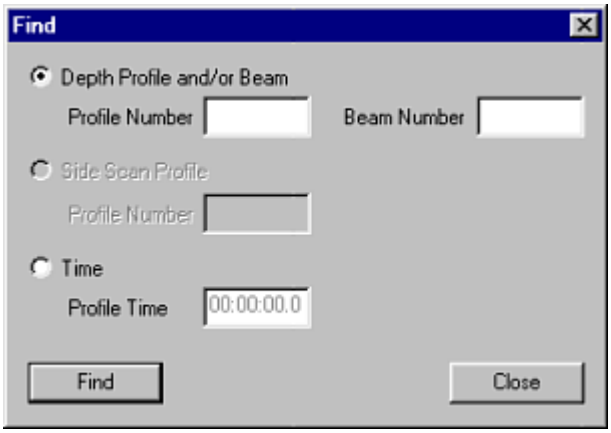
Find soundings

Search a survey line for a specific swath or beam.

Menu	Edit > Find
Tool	
Pop-up	Find
Key	<Ctrl+F>

1. Select the Find Soundings command, or right-click and select **Find** from the pop-up menu in the Plan View.

The Find Soundings dialog box is displayed.

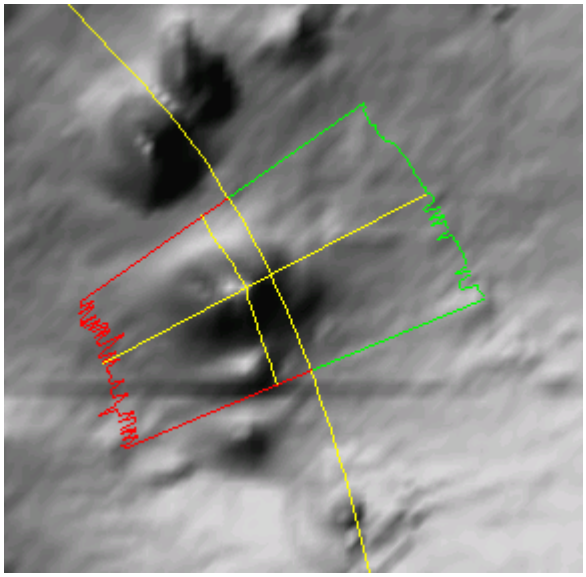


2. To search by the swath or beam number, select the *Profile and/or Beam* option.
3. Type the swath or beam number in the *Profile Number* or *Beam Number* field.
4. To search for a swath that was recorded at a specific time, select the *Time* option.
5. Click inside the *Profile Time* box and type the time in the relevant fields (hour:minutes:seconds:decimal fractions of a second). Make sure you use the 24-hour clock.
6. Click **Find**.

The selected beam or swath is highlighted in the Plan, Rear and Side Views.

Swath Coverage Outline

The swath coverage outline is a marker in the Display window that shows the location of data currently visible in the Swath Editor (see example below) The display of the outline is controlled from the Options dialog box (see "SENSOR EDITORS" ON PAGE 87).



Automatic Recentre

The automatic recentre option ensures that when the coverage outline reaches the edge of the Display window (when scanning along the track line), the display redraws with the coverage outline in the centre. This options is available in the General Options dialog box. (see "GENERAL" ON PAGE 75).

Changing Track Lines

After examining a track line, you may want examine another line.

There are four options for navigating between track lines in the Swath Editor:

- Select the Next Line command

Menu	Select > Next Line
Tool	

- Select the Previous Line command

Menu	Select > Previous Line
Tool	

- Click a line in the Display window so that the line is opened in the Swath Editor at that location
- Select a track line file name Project data tree in the Control window.

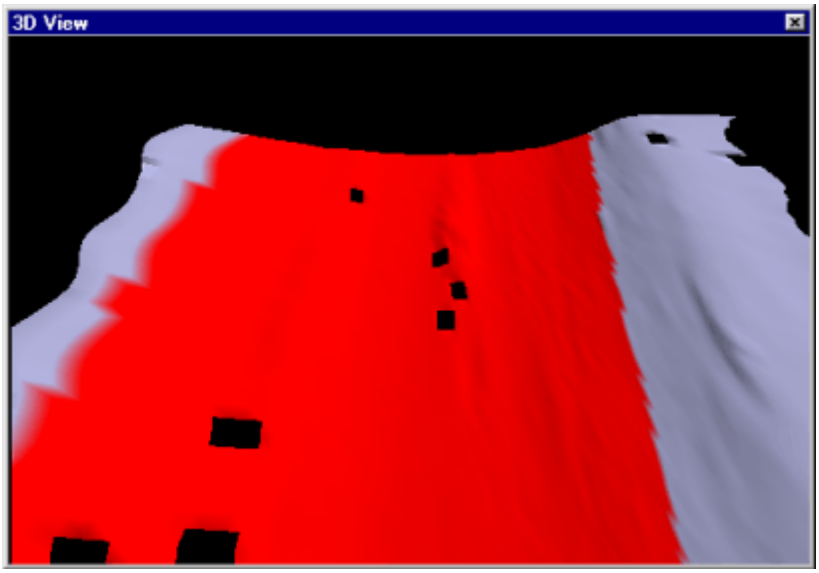
Examine 3-D Data

Swath data displayed in the Plan View can be shown in 3D format. The 3-D feature provides greater flexibility for examining and cleaning data.

Menu	Tools > Swath Editor > 3D View
Tool	

1. Select the 3-D View command.

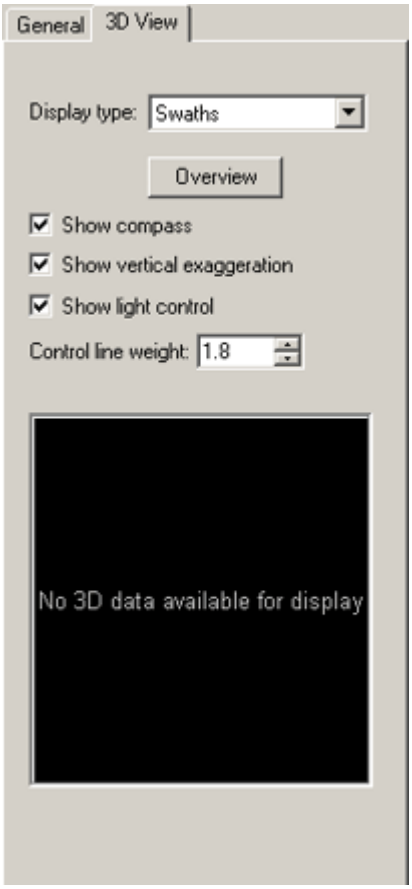
The 3-D view is displayed in the Swath Editor.



The 3-D display is manipulated by three controls:

- the compass
- the vertical exaggeration slider
- the light control.

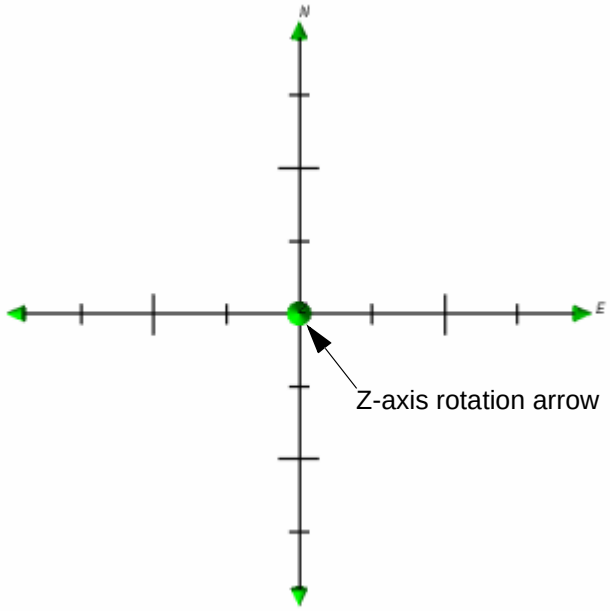
The display of data and 3-D controls is handled through the 3-D View tab, shown below.



The line thickness of these controls can be altered by changing the *Line Weight* value in the 3-D View tab.

Compass

Rotation along the X-Y-Z axis is handled by moving the arrow heads at the ends of the compass. The compass is visible in both the 3-D View and the 3-D View tab.



1. To rotate the image along the Z-axis, position the cursor on the middle arrow head, press and hold the mouse button, and drag the cursor so that the image is rotated. The compass can be rotated 90º.
2. To rotate the image along the X-Y axis, position the cursor on any of the other arrow heads, press the mouse button and drag the arrow head in a left-right or up-down direction. The compass can be rotated 360º.
3. Alternatively, the image can be rotated in the horizontal plane by grabbing the Z-arrow, then pressing the left-mouse button and moving the cursor up and down. Rotation around the Z-axis can be performed by pressing the <Shift> key and moving the cursor left and right in these windows.

The image is rotated in the same direction as the arrow heads.

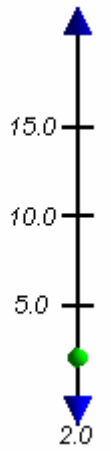
4. To return the image to its original position, click **Overview** in the 3-D View tab.

The image is restored to its original position.

Vertical Exaggeration Slider

By changing the vertical exaggeration, you can display seabed features more clearly.

Vertical exaggeration is controlled by the scale slider in the 3-D View. The slider can be displayed or hidden by selecting (or clearing) the *Show Vertical Exaggeration* check box in the 3-D Controls tab.



1. To change vertical exaggeration, position the cursor over the ball on the vertical exaggeration scale.
2. Press and hold the mouse button, and move the ball up or down the scale.

You can also change the vertical exaggeration using a dialog box.

3. Double-click the ball in the vertical exaggeration slider.

The Vertical Exaggeration dialog box is displayed.

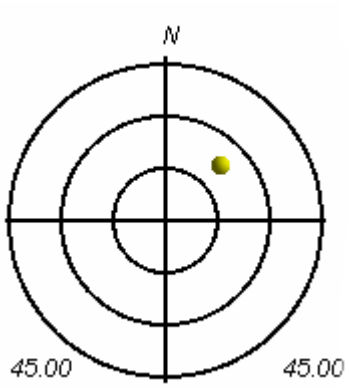
4. Type a value in the Vertical Exaggeration field and click **OK**.

The display is refreshed to show the image according to the selected exaggeration scale. The scale value is displayed below the slider.

Lighting Control

The lighting control handles shading effects by controlling the position and angle of the light.

The elevation value (on the bottom right of the compass) varies from 90º to 0º starting from the middle of sundial. The azimuth value (on the bottom left of the compass) varies from 0º to 360º in a clockwise direction. The lighting control can be displayed or hidden by selecting (or clearing) the *Show Light Control* check box in the 3-D View tab.



Azimuth Elevation

1. Position the cursor over the ball in the lighting control.
2. Press and hold the mouse button, and move the lighting control ball with the cursor.

As the ball is moved, the elevation and azimuth values change.

You can also change the shading effect using a dialog box.

3. Double-click the ball in the lighting control.

The Set Sun Position dialog box is displayed.

4. Type new values in the *Sun Elevation* and *Sun Azimuth* fields and click **OK**.

The image is refreshed to the new lighting angle.

Pan and Zoom

There are three pan options for the 3-D View:

- To pan along the X-Y axis: place the cursor in the 3-D View, press the middle mouse button and drag the display to a new location.
- To pan along the Z-axis: place the cursor in the 3-D View, press and hold the <Shift> key and middle mouse button and move the cursor in the window.
- To centre the 3-D View window on a selected point: position the cursor over the point and click the middle mouse button.

There are three zoom options:

- Press and hold the <Ctrl> key and right-mouse button and drag the cursor up in the display to zoom into display.
- Press and hold the <Ctrl> key and right-mouse button and drag the cursor down in the display to out of display
- Rotate the mouse wheel in a counter-clockwise direction to zoom into the display, or rotate the mouse wheel in clockwise direction to zoom out of the display.

Swath Cleaning

Sounding outliers can be rejected interactively or by applying automatic swath filters (see "SWATH FILTERING" ON PAGE 249).

This section examines how to interactively modify or query data in the Plan, Rear, Side, 3-D, and Profile Views.

Reject Data

Each sounding in the HIPS format carries several status flag switches that the software turns on and off as needed. Each sounding begins as an Accepted sounding and can be rejected for any reason during processing. A rejected sounding it can always be returned to the accepted status without loss of data.

A sounding rejected within the Swath Editor has the Rejected by Swath Editor flag turned on.

To reject a sounding, complete the following steps:

1. Select the data you want to reject.
2. Select the Reject command.

Menu	Edit > Status Flag > Reject
Tool	
Pop-up	Reject
Key	<R>

The Rejected by Swath Editor status flag is now on and the sounding may or may not be displayed depending if the rejected data is toggled to visible.
(See "VIEW REJECTED SOUNDINGS" ON PAGE 222)

Reject Swaths

In some instances it may be necessary to reject all soundings in one or more swaths. This can be done with the Reject Swaths function without having to select all of the data in the swaths.

1. Select some sounding in the swath you want to reject.
2. Select the Reject Swaths command.

Menu	Edit > Status Flag > Reject Swaths
Tool	
Key	<S>

All soundings in the swath (whether selected or not) now have the Rejected by Swath Editor status flag turned on. The sounding may or may not be displayed (see "VIEW REJECTED SOUNDINGS" ON PAGE 222).

Accept Data

Change rejected soundings back to the Accepted status.

1. Select the rejected data so it is highlighted.
2. Select the Accept command.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

The soundings are now flagged as accepted.

Designated Soundings

The Designated Sounding flag identifies the shoalest sounding in a feature. The purpose of the Designated Sounding flag is to ensure that the shoalest depths over significant seabed features are maintained in charts and other standard hydrographic products. For example, in a cluster of soundings surrounding an outcrop of rocks, the shoalest sounding among that cluster is identified as Designated.

For more information, see "DESIGNATED SOUNDINGS" ON PAGE 223.

1. Select the shoalest sounding from a cluster of soundings around a feature in the Swath Editor.

Menu	Edit > Status Flag > Designated
Tool	
Key	<D>

2. Select the Designated Soundings command.

The sounding is flagged as Designated.

Query Data

Select soundings to view status and other information.

3. Select the data you want to query.
4. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

The following data about the selected soundings are displayed in the Worksheet window. You can control the amount of data that is opened in the Worksheet window through the Query Fields dialog box (see "WORKSHEET WINDOW" ON PAGE 15).

- time of the swath profile
- d-time from the previous profile
- profile number
- beam number
- across track distance
- along track distance
- depth
- d-depth from the previous profile
- d-depth from the previous beam
- amplitude / phase detect
- quality value
- status flag
- horizontal error value
- vertical error value

Change sounding status

Once data information is displayed, you can change the status of the sounding.

1. Click the sounding data in the Worksheet window.
2. Select an Accept or Reject command.

Auto Cursor Mode

The auto cursor combines the select and Accept/Reject/Query functions into a single procedure.

Menu	Edit > Auto Cursor
Tool	

1. Select the Auto Cursor command.
2. Select a Reject/Accept/Query command.
3. Press the mouse button and draw a bounding box over the data.
4. Release the mouse button.

Middle-Mouse Button Selection

Soundings can also be selected and highlighted using the middle mouse button. This method is used to select specific swath profiles as well as beam profiles.

1. Select the General tab in the Control window.
2. Select the *Beam* and *Swath* check box to select both profiles.
3. Position the cursor over the Plan, Side or Rear Views.
4. Press and hold the middle mouse button while dragging the cursor across the soundings until you have selected the data you want to edit.
5. Right-click the mouse button and select a reject (see "REJECT DATA" ON PAGE 317), accept (see "ACCEPT DATA" ON PAGE 318), or query (see "QUERY DATA" ON PAGE 320) command.

The Refraction Editor

Since it is not possible to completely control the sound speed variations in water we apply sound velocity profiles to correct for this. However, refraction artifacts may still remain, especially if incorrect or insufficient sound velocity profiles are applied either during acquisition or during post processing.

To correct for this, you could choose to limit the usable swath angle (and therefore survey more lines), or you can use the Refraction Editor, which simulates the effects of altering the SVP to derive a better refraction solution.

Since refraction errors are most pronounced in the outer parts of the survey line coverage, they tend to create characteristically curved (upward or downward) refraction artifacts.

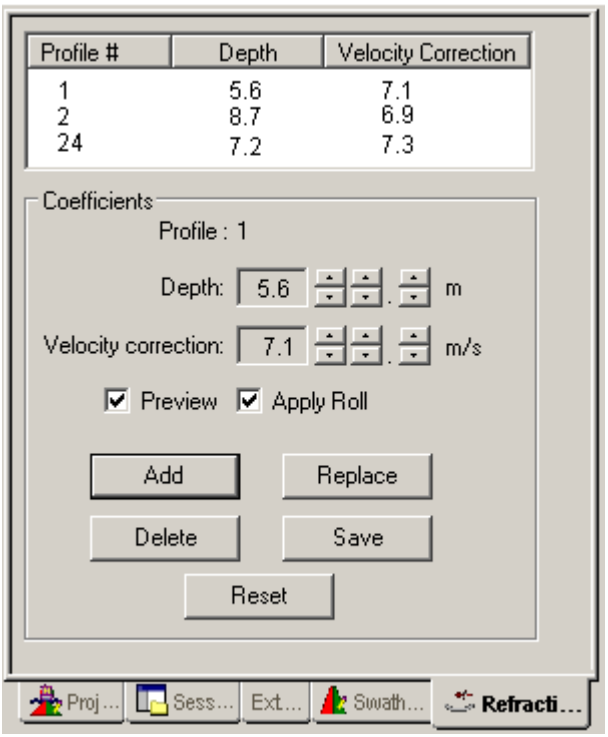
Apply the velocity correction at a user-defined depth. As you adjust the values, you can observe the changes in the shape of the swath profiles. When you arrive at the values which give you the desired correction, you can save these coefficients. Later, during the Merge process, you can apply these Refraction Editor coefficients that are recorded here.

To open the Refraction Editor,

1. Open the Swath Editor, then select the Refraction Editor command.

Menu	Tools > Swath Editor > Refraction Editor
Tool	

The Refraction Editor dialog box is displayed in the Control window.



As you change the values in the dialog box you will be able to see the effects of these changes in the Swath Editor display.

2. Select the *Preview* check box to be able view the effects of your changes.

If the *Preview* option is selected, you can also enable the *Apply Roll* option. When you first open the Refraction Editor, this control is set to be the same as the setting in the Roll section of the Vessel file (HVF). This control allows you to quickly observe the effects of applying (or not applying) dynamic roll to the sounding data.

NOTE: If there is no roll data, or if the appropriate Roll section cannot be found in the HVF, or if the survey line has already been sound velocity corrected, toggling the "Apply Roll" switch will have no effect.

3. Select the Apply Roll check box to see the effects of roll on the data.
4. Adjust the values for velocity correction and depth until the desired effect is observed.
5. Click **Add** to add the new entry to the list of coefficients.
6. To change an existing coefficient, adjust the settings and click **Replace** to update the list.
7. To remove an entry from the list select it and click **Delete**.
8. Click **Save** to save your entries.

The velocity correction and depth are recorded with the current swath profile number. While paging through the track line during cleaning, new swath/correction/depth values can be recorded.

Interpolation

When you add the first Refraction Coefficient, the correction starts at that profile and goes to the end of the line with that same correction. If you add a second Refraction Coefficient after the first, the correction applied between the two profiles is actually interpolated. This is done so that there is no “jump” in the profile when the correction at the second Refraction Coefficient is applied to the data.

This interpolation is linear. That is, if the selected profile is between two refraction entries, the depth and delta velocity are both interpolated from these entries.

Querying

Querying the data in the Swath Editor will always give the results from the display, so if you change the display with the Preview option of the Refraction Editor turned on, the query will return the results of the change.The Observed Depth data (across-track, along-track and depth) does not change.

The effects of the Refraction Editor can be turned on or off using the Calibration Module (see “CALIBRATION” ON PAGE 481). The effects of the Refraction Editor on the final merged depths can be modified any time by editing the coefficients and re-merging. The

effects can be removed by re-merging and choosing not to apply the coefficients.

13

Single Beam Editor

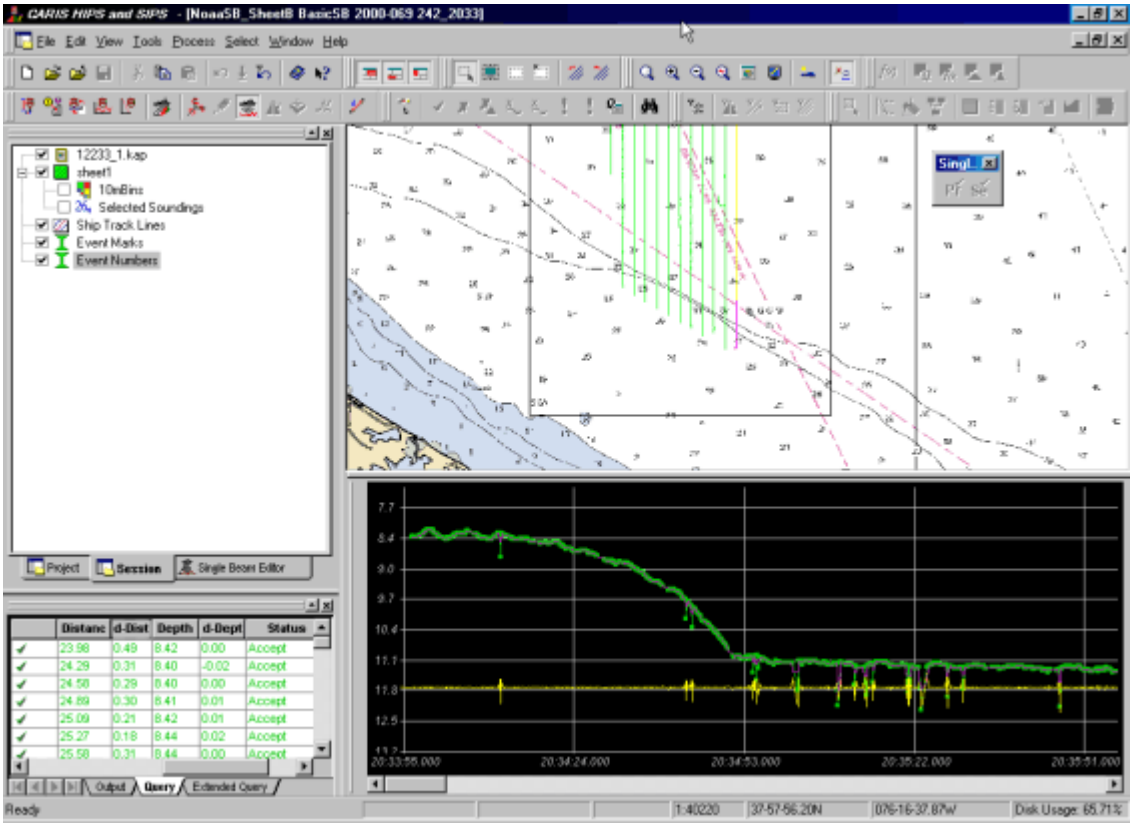
Introduction

Open the Single Beam Editor

Menu	Tools > Single Beam Editor
Tool	

1. Select a survey line in the Display window.
2. Select the Single Beam Editor command.

The Single Beam Editor is opened in the HIPS interface.



The Single Beam Editor is located (by default) below the Display window. The editor can be undocked and moved to another location like other windows in HIPS.

The display and events options are controlled from the Single Beam Editor tab in the Control window.

The soundings are shown in the editor as dots connected by lines. The horizontal scale represents distance or time. Distances are are derived from the navigation data. The first sounding with an associated position is considered the start of the line. All subsequent distances are computed from this point using sounding positions interpolated from navigation data.

When the horizontal scale is by time, the sounding timestamps are used.

You can move up or down the track line by clicking the scroll bar or turning the mouse wheel (if available).

Toolbar

The Single Beam toolbar is automatically displayed when the editor is opened. This toolbar is used for selecting primary or secondary data as Selected data (see "PRIMARY AND SECONDARY DATA" ON PAGE 329). The toolbar can moved and docked in the interface like other toolbars (see "MODIFY EXISTING TOOLBARS" ON PAGE 25).

Display Options

Change horizontal scale

The range of data that is visible along the horizontal scale can controlled by distance or time. As you increase these values more data is displayed in the editor.

The purpose is to provide a scaling method that simulates the echo sounder paper speed controls so that the digital profiles can resemble the analogue records.

1. Make sure the Control window is open and that the Single Beam Editor tab is displayed.
2. Select either the *By Time* or *By Distance* options.
3. Select a range of data by clicking the *Horizontal Scale* arrow buttons.

Adjust vertical scale

The vertical scale displays the distance values. The *Auto* setting uses the maximum and minimum soundings in the current view to set the depth values for that view. As you move along the track line, the depth values change to accommodate the visible sounding data.

You can set a scale as a ratio of the vertical scale to the horizontal scale. As the ratio is increased, the vertical display of data becomes more exaggerated.

To set a vertical scale:

1. Clear the *Auto* check box.
2. Click the *Vertical Scale* arrow buttons to select an appropriate scale.

The display in the Single Beam Editor View is refreshed to accommodate the selected scale.

Adjust horizontal labels

The horizontal labels show the depth or time values at regular intervals along the survey line. The *Auto* selection spaces the labels far enough apart so you can see them clearly. You can determine the spacing you want for the labels.

1. Clear the *Auto* check box in the *Horizontal Labels* section.
2. Click the arrow buttons to select a new scale for the labels. If you select *By Distance* for the horizontal scale, then the labels are displayed as metres/feet. If you select *By Time*, the labels are displayed hours/minutes/seconds/fractions of a second.

The display in the Single Beam Editor is refreshed to show the labels according to the selected settings.

Events

Event marks are commonly generated by the logging system at periodic intervals. Each event mark contains a unique identifier and time stamp.

Display all event marks in a table.

1. Select the Event tab in the Control window.
2. Select the *Show Event Marks* check box.

The event marks are displayed in a table format.

View moving average data

You can see the moving average data in the Single Beam Editor by selecting any of the following check boxes in the Single Beam Editor tab in the Control window. For more information on applying Moving Average filtering to single beam data, see "SINGLE BEAM CLEANING" ON PAGE 262.

- *Data* graphs the raw sounding data. This box is checked by default.
- *Moving Average* graphs the computed moving average from the adjacent data points as described above.
- *Difference* graphs the difference between the original data values and the moving average for each data point.
- *Thresholds* displays horizontal lines in the graph indicating the difference values that are rejected during filtering. Threshold is a multiple of the standard deviation (sigma).

Primary and Secondary Data

Dual frequency data can be imported through the Generic Data Parser, the Hypack converter and the Winfrog converter.

These frequencies (and the associated data) are designated as Primary or Secondary. By default, all Primary frequency data is designated as Selected. Selected data can be processed further in HIPS and other CARIS products.

Show/Hide data

1. Make sure the Single Beam Editor tab in the Control window is displayed.
2. Select or de-select the following check boxes to show or hide data. When the box is checked, the sounding data associated with that frequency is visible in the editor.
   - *Primary*
   - *Secondary*
   - *Selected*

The Single Beam Editor is refreshed to show the selected frequencies.

Select Primary/ Secondary data

Although Primary data is automatically designated as Selected, you can, if needed, flag the Secondary data as Selected data. The two frequencies cannot be averaged.

1. Make sure that all frequency data is visible in the editor by selecting the *Primary*, *Secondary*, and *Selected* check boxes.
2. Press and hold the mouse button and drag the cursor to highlight both the Primary and Secondary data.
3. Select a Primary or Secondary command.

Menu	Edit > Single Beam > Primary/Secondary
Tool	
Pop-up	Primary/Secondary
Key	<1>/<2>

The highlighted Primary or Secondary data is now marked as Selected.

Change Sounding Depths

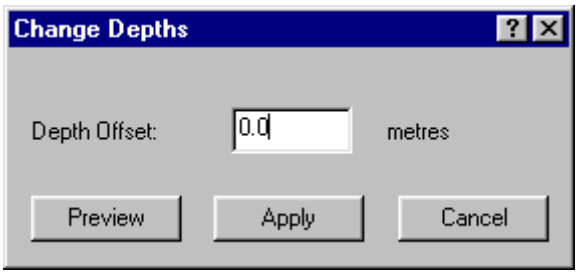
The Single Beam Editor lets you modify the digital depths created by the echo sounder's digitizer. Sometimes when fish or other obstructions in the water column trigger the digitizer, the sea bottom may still be visibly detectable on the analog sounding graph. This function lets you set the echo sounder's depth values to the analog sounding depth values.

Change Depths

1. Select the *Selected* check box, if needed.
2. Press and hold the mouse button, and drag the cursor to highlight the Selected soundings.
3. Select the Change Depths command.

Menu	Edit > Single Beam > Change Depths
Pop-up	Change Depths

The Change Depths dialog box is displayed.



The *Depth Offset* value is always set at 0.0. To move the sounding to a deeper depth level, enter a positive value (for example, 5.5); to move the sounding to a shallower depth level, enter a negative value (for example, -5.5).

4. Click **Preview** to view the sounding at the new level.
5. Click **Apply** to set the sounding at the new depth and close the dialog box.

You can also change depth values directly in the editor. This option omits the preview function of the dialog box.

1. Press and hold the mouse button and drag the cursor to highlight the Selected data sounding.

When the mouse is positioned over the selected data, the cursor becomes a two-headed arrow with a Z symbol beside it.

2. Press and hold the mouse button, and drag the cursor either up or down.

The highlighted soundings move with the cursor.

3. Release the mouse button when the soundings are at the new depth.

Add Depths

Add a new sounding by supplying time and depth. The position of the new sounding is interpolated from the existing navigation data (using the supplied time). A quality code can also be entered. If you enter a new depth, a new observed depths file and a new slant range (travel time) file is generated.

You can add depths by selecting an existing sounding and inserting a new depth or by creating a new time stamp without selecting an existing sounding.

1. Select the Add Depths command.

Menu	Tools > Single Beam Editor > Add Depths
Pop-up	Add Depths

The Worksheet window is displayed, if it is not already open.

2. Select the *Time* field and type a new timestamp value.
3. Select the *Depth* field and type a new depth value.
4. Select a new quality flag, if needed.
5. Press <Enter> to add the new depth value to the file, or press <Esc> to undo the change.

Single Beam Cleaning

Sounding data can rejected interactively or by applying the single beam filters (see "SINGLE BEAM CLEANING" ON PAGE 262).

This section covers how to interactively reject, accept and query data in the Single Beam Editor. These processes are similar to the Swath Editor.

Reject Data

Each sounding carries several status flag switches that the software turns on and off as needed. Each sounding begins as an Accepted sounding and if rejected for any reason during processing it can always be returned to the accepted status without loss of data.

A sounding rejected within the Single Beam Editor has the Rejected by Swath Editor flag turned on (the same status flag is used for both swath and single beam data).

To reject a sounding, complete the following steps.

1. Select the data to be rejected so it is highlighted.
2. Select the Reject command.

Menu	Edit > Status Flag > Reject
Tool	
Pop-up	Reject
Key	<R>

The Rejected by Swath Editor status flag is now on and the sounding may or may not be displayed depending if the rejected data is visible.

Show/Hide rejected data

You can show or hide rejected data. When the rejected data is turned off, these soundings are no longer visible in the Single Beam Editor.

Menu	View > Show Rejected
Tool	
Key	<F4>

1. Select the Show Rejected command.

When the rejected soundings are visible, the display is adjusted accordingly.

Accept Data

Change rejected soundings back to the accepted status.

1. Select the rejected data so it is highlighted.
2. Select the Accept command.

Menu	Edit > Status Flags > Accept
Tool	
Pop-up	Accept
Key	<A>

The soundings are now flagged as accepted.

Query Data

View data on selected soundings.

1. Select the data to be queried.
2. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

The following data is displayed in the Worksheet window. The data is colour coded depending on whether it is Primary, Secondary or Selected frequency data.

- Record
- Type (primary, secondary, or selected)
- Timestamp
- d-Time
- Approximate Position
- Distance
- d-Distance
- Depth
- d-Depth
- Status

Change sounding status

Once data information is displayed, you can change the status of the sounding.

1. Select the sounding data in the Worksheet window.

2. Select Accept or Reject commands.

Change sounding position

You can also change a sounding's position using the Worksheet window.

1. Select a sounding in the Worksheet window so it is highlighted.
2. Double-click the depth value.
3. Type a new depth value.
4. Click away from the highlighted area to de-select the sounding data.

Auto-Cursor Mode

The auto cursor combines the select and Accept/Reject/Query functions into a single procedure.

Menu	Edit > Auto Cursor
Tool	

1. Select the Auto Cursor command.
2. Select a Reject/Accept/Query command.
3. Press and hold the mouse button and drag the cursor over the data so it is highlighted.
4. Release the mouse button.

The data is marked as rejected/accepted or it is queried.

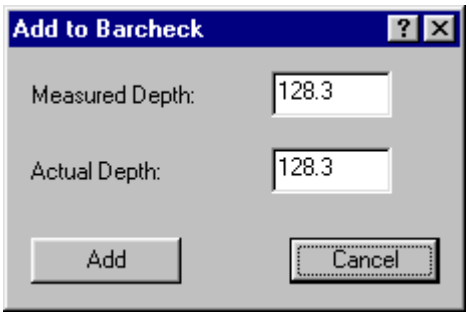
Barcheck Function

The barcheck function lets you generate a Sound Velocity Profile (SVP) file based on a list of measured depths against a list of actual depths. The file can then be used to perform sound velocity corrections.

1. Make sure the Selected data is visible.
2. In the Single Beam Editor, press and hold the mouse button and use the cursor to highlight the sounding(s).
3. Select an Add to Barcheck command.

Menu	Edit > Single Beam > Add to Barcheck
Pop-up	Add to Barcheck

The Add to Barcheck dialog box is displayed.



- The *Measured Depth* field displays the depth that was recorded by the sounder.
- The *Actual Depth* field displays the depth that was recorded by the barcheck.

If more than one sounding is selected, then a mean is calculated from the selected soundings.

4. Enter new depth values, if necessary.
5. Click **Add**.
6. Repeat Steps 2 to 5, as needed.
7. Click the Barcheck tab in the Control window.

The table shows the Measured Depth, the Actual Depth and the sound velocity.

8. If you want to delete a section, highlight it and click **Remove**.
9. Click **Save**.

The Save dialog box is displayed. The default location for the file is ..\Hips\Svp.

10. Type a name for the file and click **Save**.

14

Tide Editor

Introduction

The Tide Editor presents information from a selected tide file in both graphical and tabular formats. You can use the editor to edit existing data or to create a new tide file, to apply to survey lines before the Merge process.

Tide files are located by default in …\Hips\Tide and have a .tid file extension.

1. Select the Tide Editor command in the HIPS and SIPS interface.

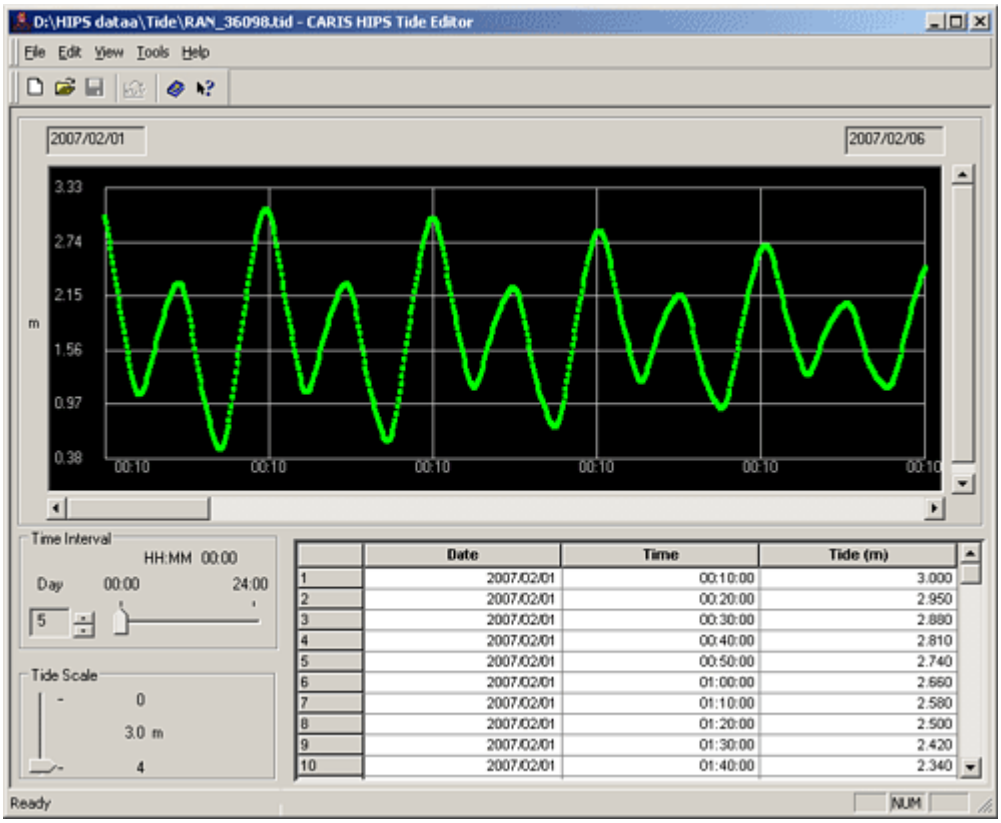
Menu	Edit > Tide Editor
Tool	

The Tide Editor is displayed.

1. Select the Open File command to display the Open dialog box.
2. Select a tide file and click **Open**, or type the file name and path in the field.

Menu	File > Open
Tool	
Key	<Ctrl+O>

The selected tide file is displayed in the editor. The graphical display shows a tide versus time graph of the data. Tide-level values are represented on the vertical axis; time is represented on the horizontal axis. The tide data is also displayed in a table below the graphical display.



Adjust tide scale

To control the vertical scale of the graphical display:

- Move the *Tide Scale* slide bar to select a range for the height of the vertical tide axis.

Adjust time interval

Change the horizontal scale by adjusting the *Time Interval* setting.

- The day option lets you view data by 24-hour increments.
- The hour/minute option lets you view data in increments of less than 24 hours.

1. Click the *Day* arrow buttons to select the range of days for displaying data.
2. If the *Day* value is set to zero then move the slide bar to select the minute/hour range for the display of data.

The display of tide data is adjusted for the new time interval along the horizontal axis.

Save a tide file

Save changes to an existing tide file.

1. Select the Save command.

The file is saved with the changes.

Menu	File > Save
Tool	
Pop-up	<Ctrl+S>

Editing Tide Data

Edit table data

Edit data in an existing tide file.

1. Select a Date/Time/Tide cell in the tide table.

The cursor is displayed in the cell.

2. Remove the existing data and type a new entry.

The graphical display refreshes to show the new data.

Delete a tide entry

1. Press and hold the mouse button while dragging the cursor over data in the graph, or click the appropriate number cell in the tide table.

The data is highlighted.

2. Select the Delete command.

Menu	Edit > Delete
Key	<Delete>

The tide data is removed from the graphed line and the table.

Search time gaps

Tide gauges, recording tidal height observations, may be subject to occasional outages resulting in gaps in the tidal time series. The time gaps option lets you search for these gaps. You can enter new data, if needed.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.

2. Select the General tab if it is not already displayed.
3. Enter a value (in minutes) in the *Time Gaps* box.
4. Click **OK**.
5. Select the Next Time Gap command.

Menu	Tools > Next Time Gap
Tool	
Key	<Ctrl+G>

When the interval (in minutes) between adjacent data points is the same as, or greater than, the *Time Gaps* value, the adjacent data points are highlighted.

6. Repeat the process, if necessary, by selecting the Next Time Gap command for each tide data point on the graphed line.

You can enter new data between two values.

Enter new data

Enter data for a new or existing tide file.

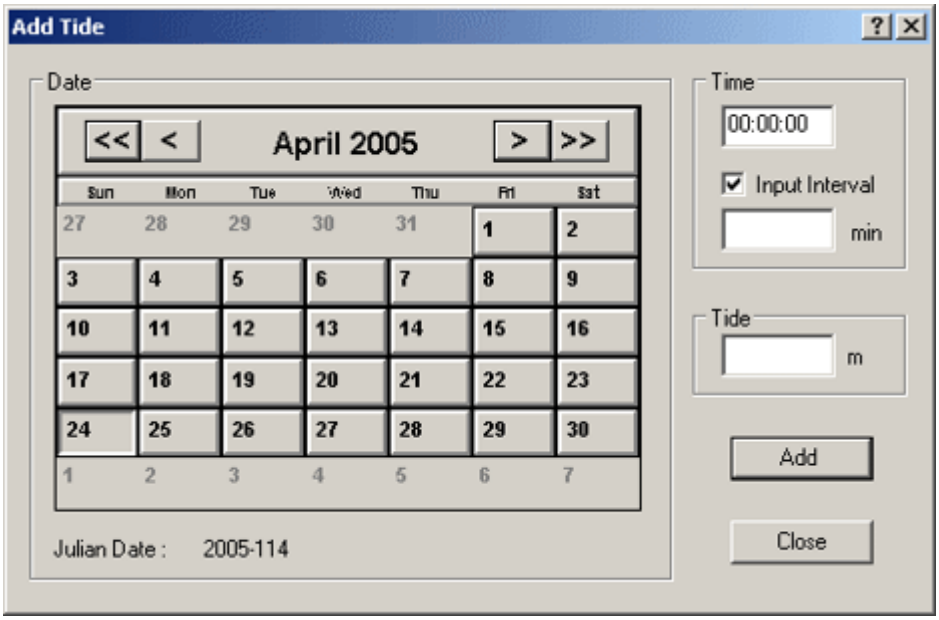
1. Select the Add Tide command.

Menu	Edit > Add Tide
Key	<Insert>

The Add Tide dialog box is displayed.

2. Change the date (if necessary) by clicking the arrow buttons at the top of the calendar until you have selected the desired month and year.
3. Click the appropriate day button on the calendar.

The selected date (year and day) is displayed below the calendar.



The default time is the time of the currently selected tide record.

4. Click any of the three sections (00:00:00) in the *Time* box so they are highlighted and type the desired time (use the 24-hour clock).

The *Input Interval* option lets you quickly enter a series of tide values at a regular time interval, using the <Enter> key. You do not need the mouse to move the cursor between fields.

5. If you want to regularly increment the times for tide data, click the *Input Interval* check box.
6. Enter the minutes that the time value is to be incremented.
7. Enter a tide value.
8. Click **OK**.

The new tide data is displayed in the editor. If you selected the Input Interval option, the time is incremented by the value set in the *Input Interval* box.

Create a New Tide File

You can create new tide file.

1. Select the New command.

Menu	File > New
Tool	
Key	<Ctrl+N>

Any tide file that is currently open in the editor is closed.

2. Select the Add Tide command and type a new tide entry (see “ENTER NEW DATA” ON PAGE 340).

The data is displayed in graph format in the editor.

3. Repeat until all values are entered.

4. When ready, select the Save or Save As command.

Menu	File > Save/Save As
Tool	
Key	<Ctrl+S>

The Save As dialog box is displayed.

5. Type a name for the file and click **OK**.

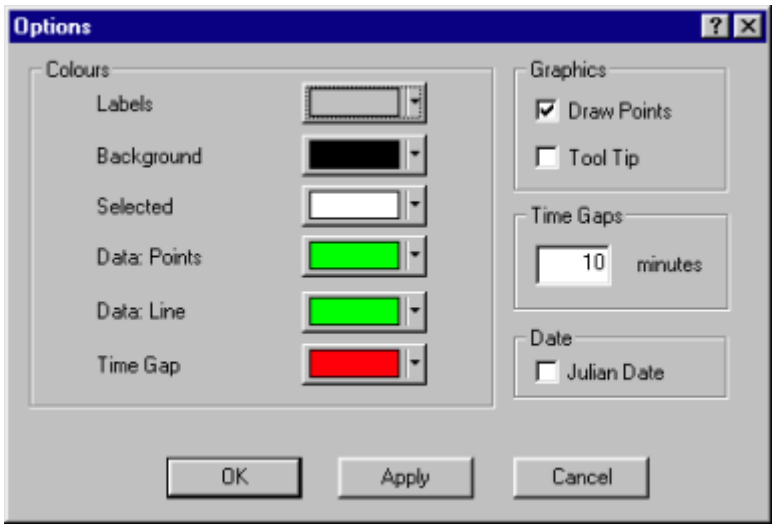
Options

You can adjust the colour display of graphical data.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.



Colour

2. Select a colour from the drop-down colour palettes, or create a custom colour from the standard Windows colour picker.
3. Click **Apply** to apply and save the settings for the project.
4. Click **OK** to close the dialog box.

The display is now refreshed to show the selected colours.

Show/Hide tide points

1. Select the Options command.

The Options dialog box is displayed.

2. Select the *Draw Points* check box.

When the Draw Points check box is selected, the data points are visible.

3. Click **Apply** to apply and save the settings for the project.
4. Click **OK** to close the dialog box.

Show/Hide tool tips

A tool tip is a label that displays a specific tide level on the graph, when the cursor is positioned over it. You can show or hide tool tips.

1. Select the Options command.
2. Select the *Tool Tip*s check box. If this option is checked then it is enabled.
3. Click **Apply** to apply and save settings for this project.
4. Click **OK** to close dialog box.

Set date display

The date in tide file table can be displayed in either Julian Day format or regular calendar format.

1. Select the Options command.

Menu	Tools > Options

The Options dialog box is displayed.

2. Click the *Julian Date* box, if the box is checked, this option is selected.
3. Click **Apply** to apply and save settings for this project.
4. Click **OK** to close dialog box

Show/Hide toolbar

Display or hide the toolbar.

1. Select the Toolbar command.
2. Click the Toolbar menu item. When the check mark is displayed, the toolbar is visible.

Menu	View > Toolbar

Toggle status bar

Display or hide the status bar.

1. Select the Status Bar command.
2. Click the Status Bar check box. When the check mark is displayed, the status bar is visible.

Menu	View > Status Bar

Exit the Tide Editor

Menu	File > Exit

Close the Tide Editor.

1. Select the Exit command.

You are prompted to save any unsaved changes.

2. Click **Yes** to save the changed file.

The Tide Editor is closed.

15

Field Sheets

Overview

The field sheet system for HIPS and SIPS is used for organizing data products created from the cleaned and processed bathymetry and side scan data.

The field sheet consists of a data directory with a field sheet definition file and a CARIS file for storing vector products.

A field sheet can be used for the following purposes:

- to create a BASE surface
- to create a side scan mosaic
- to create a layer of tiled soundings
- to create a layer of contours from a BASE surface or a layer of tiles
- to select soundings from a layer of tiles
- to generate a profile from a BASE surface
- to create layers of mark-up data

Note: this chapter describes the creation and use of field sheets in HIPS and SIPS.

For information on using the CARIS Field Sheet Editor to add contours, suppressed soundings and other layers to field sheets, see the Field Sheet Editor User's Guide. The Field Sheet Editor is a separate program opened from HIPS and SIPS.

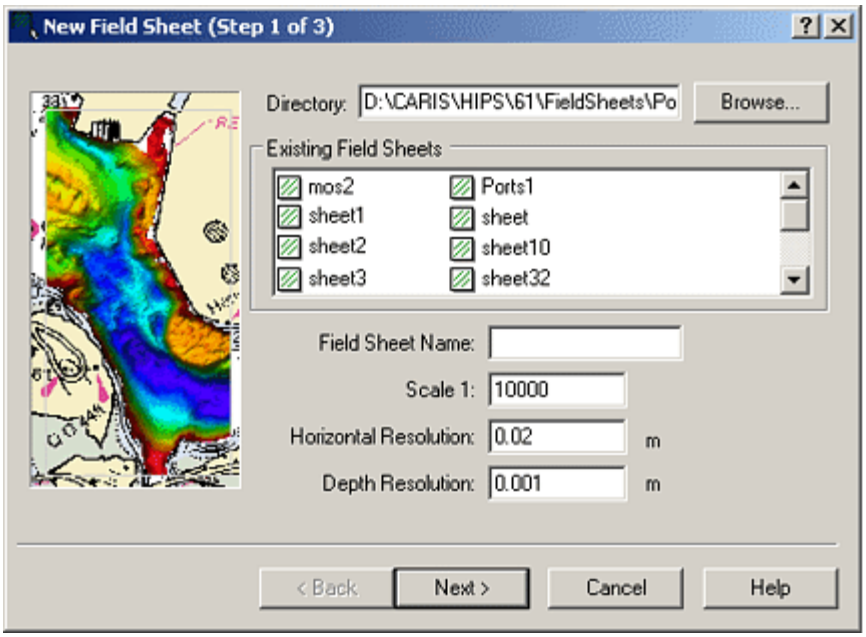
Create a Field Sheet

Menu	Process > New Field Sheet
Tool	

Define an area for a new field sheet.

1. Select the new New Field Sheet command.

The New Field Sheet (1) dialog box is displayed.



2. Type a name for the field sheet in the *Name* text box.
3. Define the scale of the field sheet.

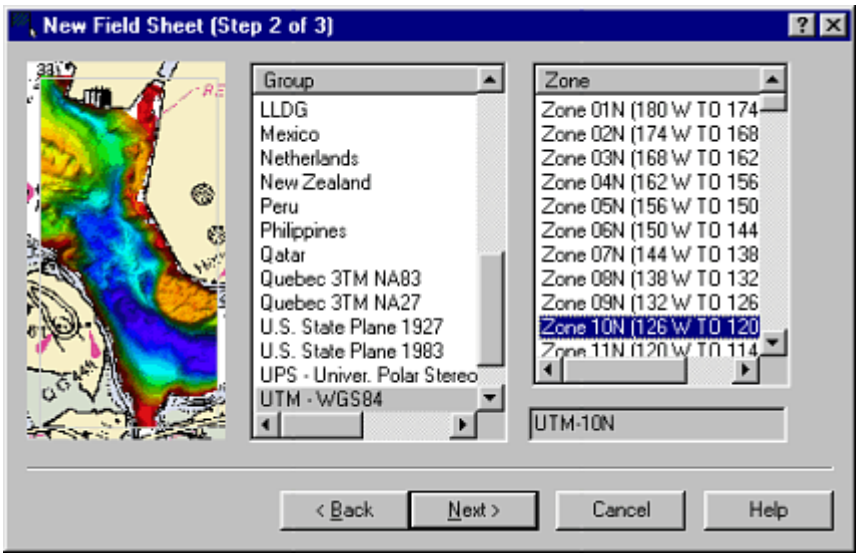
The *Horizontal Resolution* field sets the minimum distance between two adjacent coordinate values in the field sheet. The first time you create a field sheet this field will contain the default value of 0.020 (the minimum allowable resolution for S-57). You can reset this field to default to a different value.

4. Type a new value in the *Horizontal Resolution* field, if needed.

The *Depth Resolution* sets the minimum distance between two adjacent depth values. The *Depth Resolution* field contains a default value of 0.0010 (default resolution used in CARIS maps for heights/depths).

5. Type a new value in the *Depth Resolution* field, if needed.
6. Click **Next**.

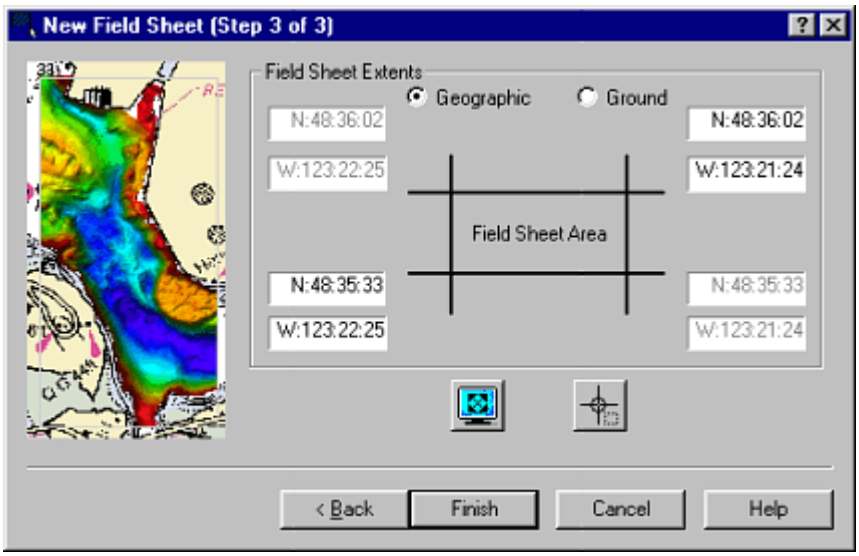
The New Field Sheet (2) dialog box is displayed.



7. Select a map projection and zone for the field sheet. The pre-defined map projection parameters are located in ..\Hips\System\mapdef.dat. The mapdef.dat is a text file so you can append or edit the projections, as needed.
8. The *Projection Key* is automatically displayed when the Zone is selected.
9. Click **Next**.

The New Field Sheet (3) dialog box is displayed.

10. Select either the *Geographic* or *Ground* coordinates option.
11. Define the field sheet extents by one of three methods:
    - enter the geographic or ground coordinates
    - use the current extents of the Display window
    - use the mouse to create a rectangle defining the extents.



Coordinates

12. Enter the northeast and southwest extents by clicking a section in a box and typing the information; or by using the arrow keys to toggle the values into the entry fields.
13. Make sure that the Display window only displays the area that is selected for the field sheet.
14. Click the Display Window Extents button.

Tool

15. Click the Bounding Box button.

Tool

The cursor in the Display window is shown as a crosshair.

16. Press and hold the mouse button, and drag the cursor across the area where you want to create the field sheet.

A rectangular box is drawn across the area where the cursor was dragged.

- If you want to move the bounding box, position the cursor inside the box so it becomes a four-headed arrow. Press and hold the mouse button, and then drag the box to a new location.
- If you want to resize the bounding box, position the cursor on any of the box handles so the cursor becomes a two-headed arrow. Press and hold the mouse button and drag the handle to resize the box.

17. After the area for the field sheet has been defined, click **Finish**.

A new field sheet is created. The new field sheet data and files are located, by default, in ..\Hips\Fieldsheets\ProjectName\FieldSheetName.

Open and Close Field Sheets

Open a field sheet in a project or session.

If a session file is opened then all field sheets that were open when the session file was last saved are automatically opened as well.

1. Select the Open Field Sheets command.

Menu	File > Open Field Sheets

The Open Field Sheets dialog box is displayed.

2. Select an existing field sheet.
3. Click **Open**.

The field sheet outline is displayed in the Display window. The field sheet file tree is displayed in the Session tab of the Control window.

Layers produced in the Field Sheet Editor (for example, contours, soundings values or text), can be displayed by expanding the field sheet file tree and selecting a layer check box (you may have to select the Refresh command to view the data).

Set as active field sheet

It is possible to create field sheets with different coordinate systems and to have them open in the same session. But only one field sheet can control the coordinate system used in the Display window. The Set as Active Field Sheet function identifies the field sheet that controls the coordinate system in the Display window.

1. Select a field sheet file in the Control window and select the Set as Active Field Sheet command.

Menu	Process > Set as Active Field Sheet
Pop-up	Set as Active Field Sheet

The field sheet is highlighted.

1. Select a field sheet from the Field Sheets tab in the Control window.

Close all field sheets

2. Select the Close Field Sheets command.

Menu	File > Close Field Sheets

The field sheet outline is removed from the Display window and the file name is no longer displayed in the Control window.

Close a single field sheet

1. Select a field sheet file in the Control window.
2. Right-click the mouse and select the Close command from the pop-up menu.

Pop-up	Close

The field sheet is no longer open in the interface.

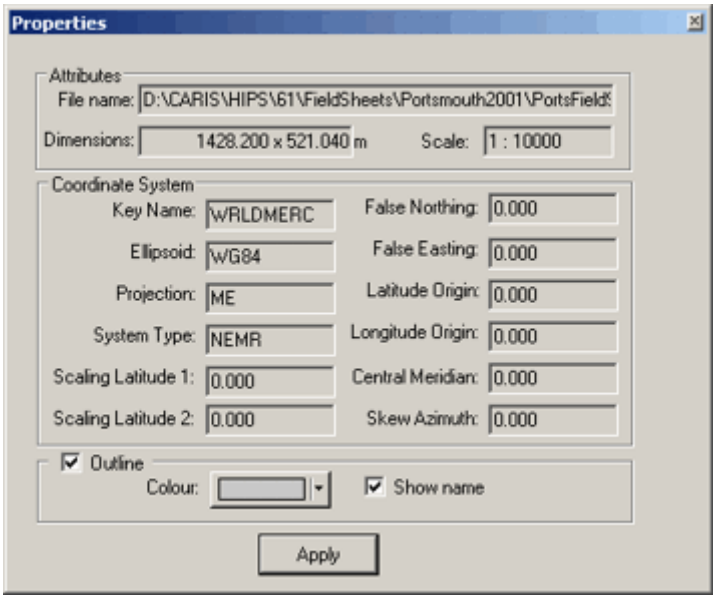
Set Field Sheet Properties

Set the display properties of a field sheet.

1. Select a field sheet layer in the Session file of the Control window.
2. If the Properties window is already not open, select the command, or right-click the layer and select Properties from the pop-up menu.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window for field sheets is displayed.



The *File Name*, *Dimensions*, *Scale* and *Coordinate System* fields are read-only and cannot be modified.

3. Select *Outline* to display a border around the field sheet area.
4. Select a border *Colour* from the colour picker, or, create a custom colour from the standard Windows Colour Palette.
5. Show the name of the field sheet in the Display window by selecting the *Show Name* check box.
6. Click **Apply**.

CARIS Field Sheet Editor

The Field Sheet Editor is a separate program opened from HIPS and SIPS.

Using the CARIS Field Sheet Editor you can add contours, suppressed soundings and other layers to field sheets. These data layers can be displayed in HIPS/SIPS by expanding the field sheet file tree in the Control window and selecting the layer check box.

For more information, see the Field Sheet Editor User's Guide or the on-line help in the application.

Open the Field Sheet Editor

1. Make sure that you have saved the project as a Session file (see "SESSION FILES" ON PAGE 159).
2. In the Session tab, select the field sheet that you want to open in the editor.
3. Select the Field Sheet Editor command.

Menu	Tools > Field Sheet Editor
Tool	

The Field Sheet Editor is open.

Tiling

Tiles are created in the Field Sheet Editor and are used in that application to create selected soundings, contours and other data layers. To learn how more about the tiling process, see the CARIS Field Sheet Editor User's Guide.

Generally, the tiling process splits a selected area into four cells (tiles) and continues subdividing each cell until two user-set parameters are satisfied:

- the minimum number of points in the tile, and
- the minimum size of the tile.

When you close the Field Sheet Editor and return to HIPS and SIPS, the you are prompted to save the tile and associated layers created in the Field Sheet Editor.

Tiles can also be created in HIPS and SIPS for performing statistical surface cleaning on sounding data (see "STATISTICAL SURFACE CLEANING" ON PAGE 457).

Open and Close Tiles

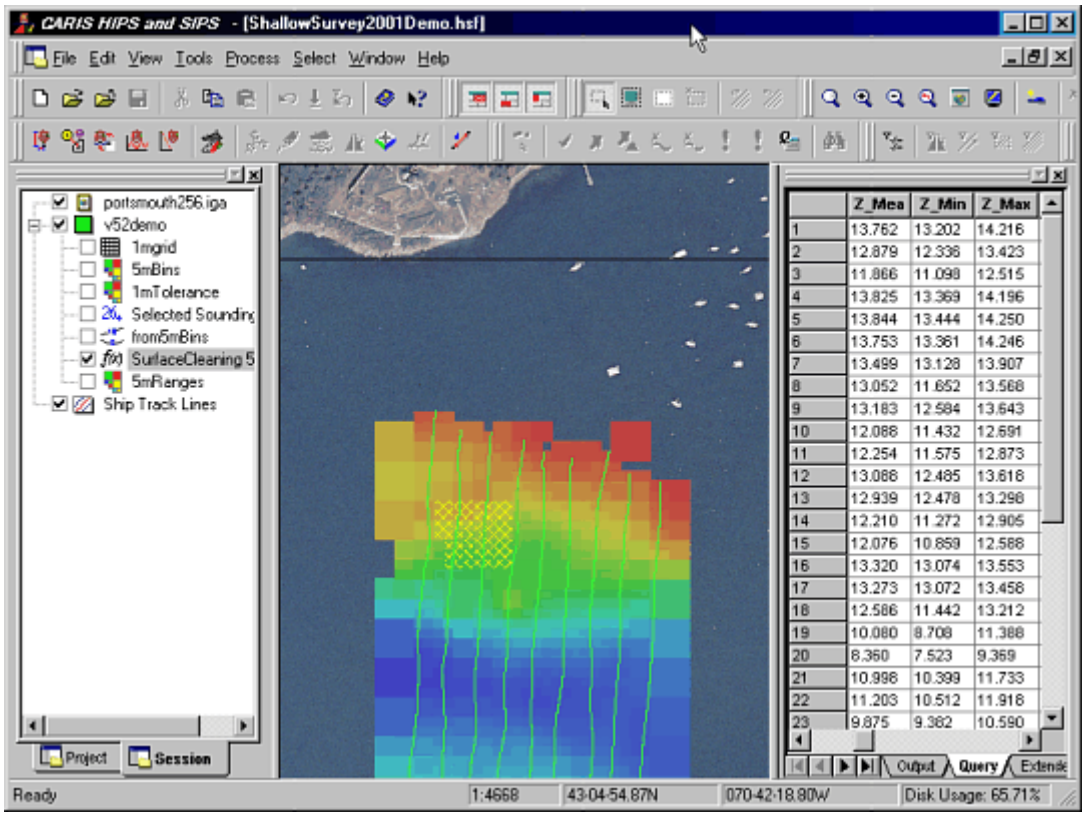
Menu	File > Open Field Sheets

1. Select the Open Field Sheets command.

The field sheet file name is displayed in the Session tab of the Control window.

2. Expand the field sheet file tree by clicking the Expand (+) icon beside the field sheet so the tile file name is visible
3. Select the check box beside the file name. If the box is selected, the tile is visible in the Display window; if the box is cleared, the tile is not visible.

Below is an example of a tiled area with all the colour and border options displayed.



Close a tile file in the Display window.

1. Select the Close Field Sheets command.

Menu	File > Close Field Sheets

All tiles associated with the field sheet are closed.

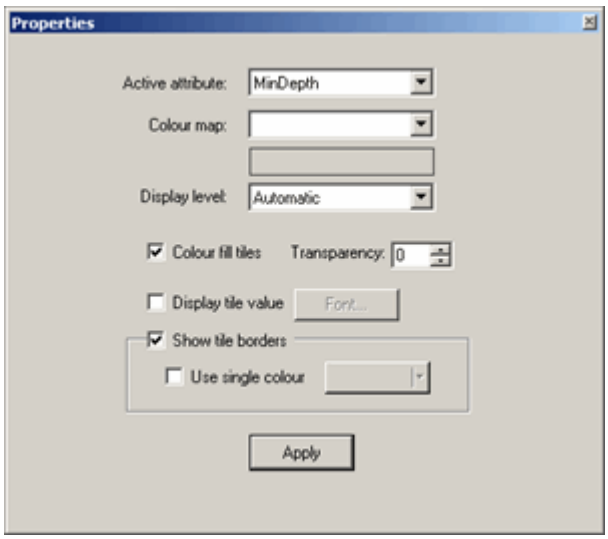
Set Tile Properties

Set the display properties for tiles.

1. Select a tile layer in the Session tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up menu.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window showing the display options for tiles is open.



The *Active Attribute* list contains all attributes included in the tile (such as the Profile number). When you select an attribute, the tile is displayed according to the attribute values

1. Select an attribute from the drop-down *Active Attribute* list.
2. Select a colour map from the drop-down list. A preview of the colour map is displayed below the list.

Change the display of tiles in the Display window.

3. Change how tiles are rendered to the Display Window by selecting from three *Display Level* options:
   - *Maximum*: Draw all tiles
   - *Size*: Draw all tiles at the selected tile size level or higher. Select a minimum tile size by clicking the arrow buttons
   - *Automatic*: Draw tiles to fit the current extents of the Display window.

The *Colour Fill* check boxes determine how attribute values are drawn in the Display window.

- If the *Colour Fill* check box is cleared, these values are drawn according to the selected colour map, but the tiles are left blank.

- If the *Colour Fill* check box is selected, then the individual tiles are filled according the colour map, but the attribute values are shown in black.

4. Select *Colour Fill* to colour code tiles according to the colour map.
5. Select a *Transparency* percentage to make features visible through the tiles.
6. Select the *Display Tile Value* check box to draw the selected attribute values.
7. Click **Font** to select a typeface and font size for the value.

The *Show Tile Border* check box determines how the border lines dividing the tile areas are displayed.

- If the *Show Tile Border* check box is cleared, the lines are colour coded according to the colour map and attribute values.
- If *Border Colour* is selected, then choose a single colour to display tile border lines.

8. Select the *Show Tile Border* check box to choose a single colour to display tile border lines.
9. Select a colour from the standard Windows colour picker; or click **Other** and create a custom colour from the palette.
10. Click **Apply** to implement the changes.

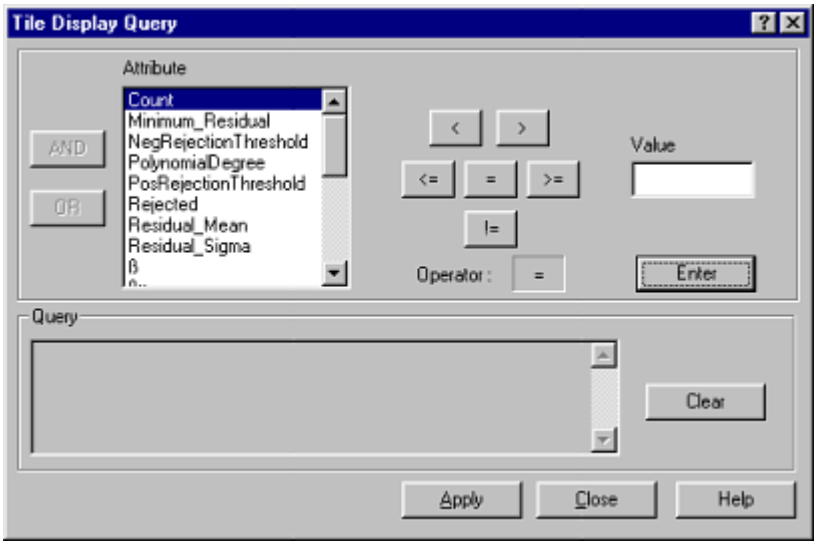
Tile Display Query

Using the Tile Display Query dialog box you can create an SQL-type query based on selected tile attributes and defined values. Only those tiles that meet the query are displayed.

1. Select the Tile Display Query command.

Menu	View > Tile Display Query

The Tile Display Query dialog box is open.



2. Select an attribute from the list.
3. Click one of the following operator buttons:
   - less than (<)
   - greater than (>)
   - less than or equal to (<=)
   - equals (=)
   - greater than or equal to (>=)
   - not equal to (!=)
4. Type a value for the attribute in the *Value* field.
5. Click **Enter**.

The search criteria you selected in the dialog box is displayed in the *Query* section.

6. To add another search criteria, click any of the following two logic operators:
   - **AND**: This operator combines two queries together so the results of both are shown in the Display window.
   - **OR**: This operator searches and displays either one of two search criteria in the Display window.

     Repeat Steps 2 through 5.

7. Enter as many search criteria as you want.
8. Click **OK**.

The Display window is refreshed so only tiles that meet the search criteria are shown.

Query Tiles

Menu	Window > Information
Tool	

1. Select the Worksheet window command if this window is not already open.
2. Make sure the tiles are open in the Display window.
3. Select the tile layer in the Session tab of the Control window.
4. Press and hold the mouse button while dragging the cursor over the tiles to be queried.

The selected tiles are highlighted.

Menu	Edit > Query Tiles
Tool	

5. Select the Query Tiles command.

The following tile data is displayed in the Worksheet window (from left to right):

- **East**: easting coordinate for the centre of the tile.
- **North**: northing coordinate for the centre of the tile.
- **Z_Mean**: mean depth value for tile.
- **Z_Min**: minimum depth value for tile.
- **Z_Max**: maximum depth value for tile.
- **Residuals_Mean**: mean value for residuals from polynomial.
- **Residuals_Sigma**: standard deviation for residuals.
- **Count**: number of points in the tile.
- **Rejected**: number of points rejected during surface cleaning.
- **NumberOfCoefficients**: the number of polynomial coefficients used.

Profiles

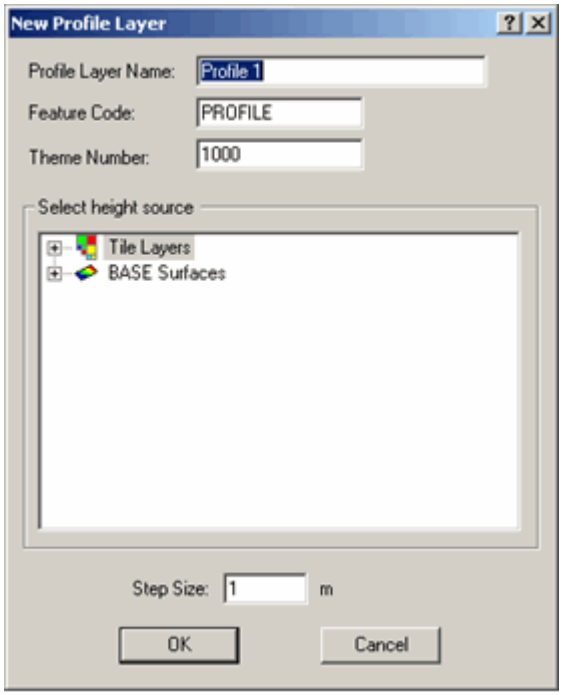
A profile is a cross-sectional view of a BASE Surface, weighted grid, or tile. The profile contains two axes: the attribute values from the grid, tile, etc. (vertical axis) and distance (horizontal axis). While in digitizing mode, the view can be rotated, zoomed or panned. Profiles are saved as field sheet layers.

Create Profile

1. Make sure that a field sheet is open and a BASE Surface or tile is visible in the Display window.
2. Select a field sheet layer in the Session tab of the Control window.
3. Select a Display Profile command.

Menu	Tools > New Profile
Tool	
Pop-up	New Profile (Control window)

The New Profile Layer dialog box is displayed.



4. Type a name for the profile layer.
5. Type a CARIS feature code for the profile. The default code is PROFILE.
6. Type a layer number.
7. Select a tile or BASE Surface attribute as a height source. To expand the file tree, click the + icon beside the layer.
8. Enter a *Step Size* value so that depth data is extracted at regular increments along the profile.

9. Click **OK**.

Digitize the profile

You can now digitize a two-point profile or multi-point profile in the Display window using the Edit Line options.

10. Click in the Display window where you want to start your profile.

Pop-up	Edit Line > Point-to-Point Line/Cubic Spline/B Spline

11. Right-click and from the Edit Line pop-up menu determine the type of line to be drawn:
    - **Point-to-Point**: Connect the points as a straight line. This is the default option.
    - **Spline**: Connect more than two points with a slightly curved line.
    - **Stream Line**: Connect more than two points by dragging the cursor to draw a line.
    - **Fitted Curve**: Connect points in a curve based on polynomials.
12. Click in another section of the BASE Surface or tile to join the points with a line.
13. Continue entering as many points as needed.
14. To remove the last entered point right-click and select Remove last.

The Profile tab in the Worksheet window displays the graphed attribute and distance data between the points. The attribute values (vertical axis) and distance levels (horizontal axis) are measured in metres or feet. You may want to expand the Worksheet window to get a better view of the data.

The profile expands as more points are entered.

To stop the profile line digitizing process

15. Right-click in the Display window and select End Line from the Edit Line menu.

The profile line is drawn in the Display window and can be selected.

Display Properties

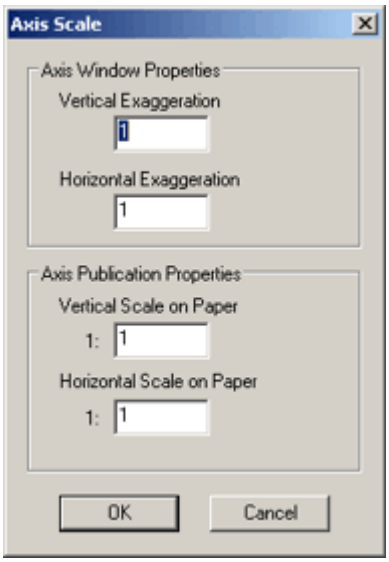
Once a profile is created you can modify its properties, including vertical and horizontal exaggeration, colour, legend display, etc.

Exaggeration

Setting the vertical and horizontal exaggeration renders a clearer display of the profile.

1. Right-click in the Profile tab and select Set Axis Scale from the pop-up menu.

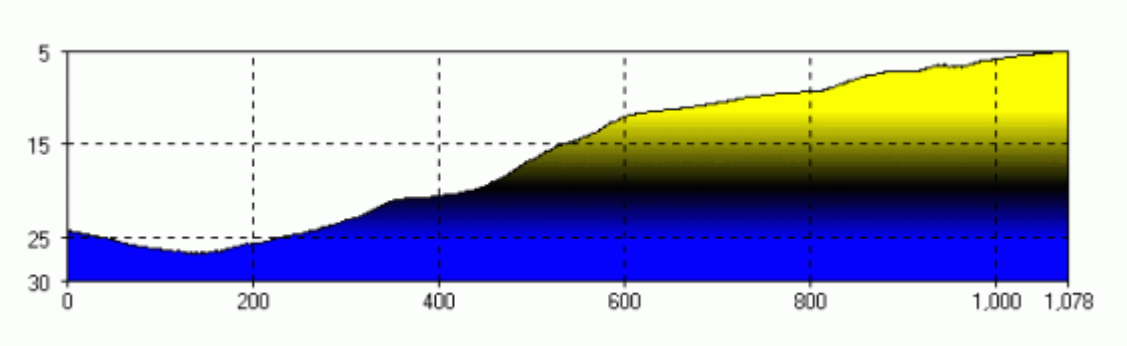
The Axis Scale dialog box is displayed.



The *Axis Window Properties* sets the profile scale in the Profile tab; the *Axis Publication Properties* sets the profile scale for printing.

2. Type the horizontal and vertical exaggeration scales and click OK.

The Profile tab is refreshed to display the profile at the new scale (see below).

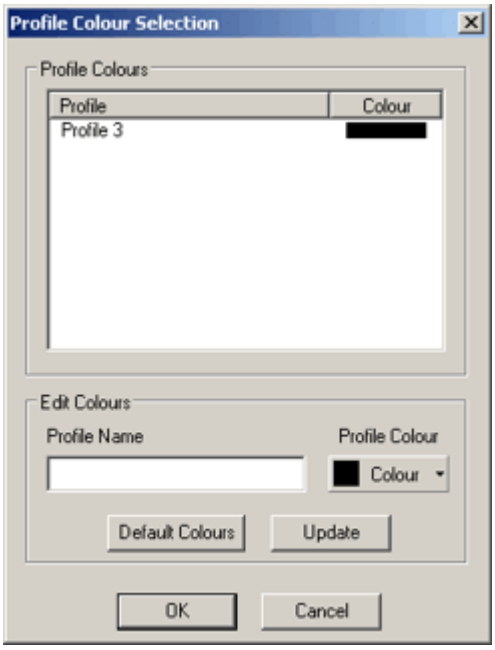


Profile Colour

Set a single colour for the profile.

1. Right-click in the Profile tab and select Profile Colour from the pop-up menu.

The Profile Colour Selection dialog box is displayed.



2. Select a profile name from the *Profile Colours* list.

The profile is shown in the *Profile Name* field.

3. Select a colour from the pull-down menu or create a custom colour using the palette.

The selected colour is shown in the dialog box.

4. Click Update.

The selected colour is shown in the *Profile Colour* list.

5. [Optional] Click **Default Colours** to reset the colour back to the previous one.

6. Click **OK**.

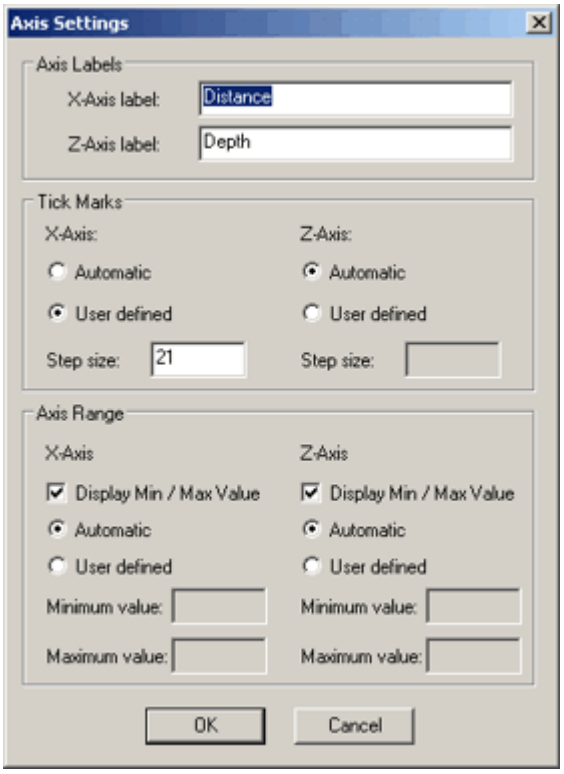
The profile is changed to the selected colour.

Axis Settings

Select the display for the X and Z axes.

1. Right-click in the Properties tab and select Graph Axis from the pop-up menu.

The Axis Settings dialog box is displayed.



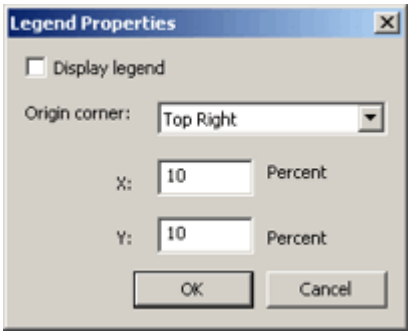
2. Type the labels for the X and Z axes in the *X-Axis* and *Z-Axis* fields.
3. Select the X-axis and Z-axis increments as *Automatic* (derived from the maximum and minimum values in the profile) or as *User Defined*.
4. If you selected *User Defined*, type the *Step Size* value in the field.
5. Select the *Display Min/Max Value* check boxes to show the minimum depth value (Z-axis) or maximum distance value (X-axis).
6. Select *Automatic* to scale data according to the minimum and maximum values in the profile; or select *User Defined* to enter scale data according to selected depth and distance values.
7. If you selected *User Defined*, type the *Minimum* and *Maximum Values* for the data range.
8. Click **OK**.

Legend

Display a legend and set the location of the legend.

1. Right-click in the Profile tab and select Legend Properties from the pop-up menu.

The Legend Properties dialog box is displayed.



2. Select the *Display Legend* check box to show the legend in the Profile tab.
3. Select the location of the legend in the tab by selecting a *Origin Corner* from the drop-down list.
4. Select an offset from the selected edge by typing the *X* and *Y* offsets.
5. Click **OK**.

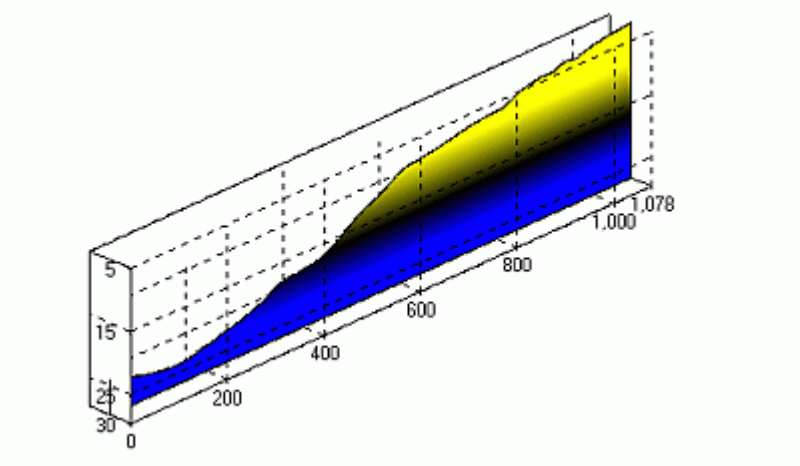
The legend is displayed in the Profile tab.

Zoom/Pan/Rotate

You can rotate, zoom, and pan the display. These options are available while in digitizing mode from a pop-up menu in the Profile tab of the Worksheet window. The profile can be rotated to a maximum of 180°.

1. Right-click in the Profile tab of the Worksheet window and select Mouse Mode > Rotate from the pop-up menu.
2. Press and hold the mouse button while dragging the cursor in Worksheet window up or down, or sideways.

The profile display is rotated according to the position where you release the mouse button.

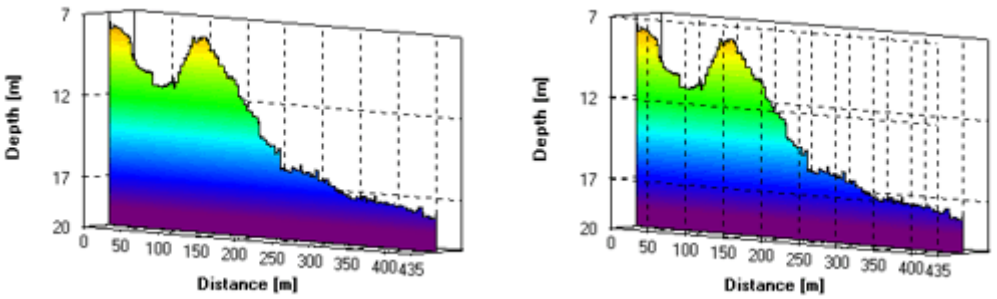


3. Zoom in or out of the profile by right-clicking in the Profile tab and selecting Mouse Mode > Zoom from the pop-up menu.
4. Pan the profile by right-clicking in the Profile tab and selecting Mouse Mode > Pan from the pop-up menu.

The zoom and pan operations require you to press and hold the mouse button while dragging the cursor in the Worksheet window.

You can view the profile with vertical and horizontal grid lines in front of profile.

5. Right-click in the Profile tab and select Wall Lines > Show Horizontal Front Line or Show Vertical Front Line from the pop-up menu.



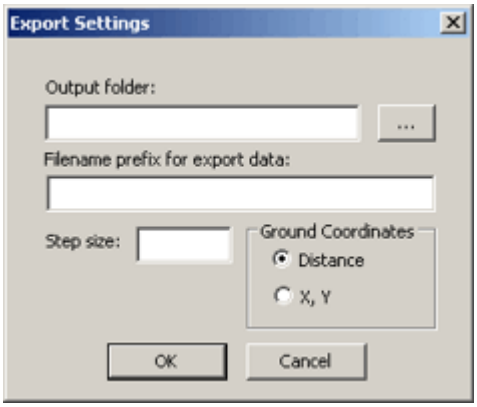
6. Select *Snap to 2D* from the pop-up menu to return the profile to its original position.

Export Profile

Create a text file containing position and depth data from a selected profile.

1. Right-click in the Profile tab and select Export Graph to ASCII from the pop-up menu.

The Export Settings dialog box is displayed.



2. Click **Browse** to select a directory for the file.
3. Type the file name prefix in the *Filename* field. All files will have the profile name appended to the end of it.
   - For example, a text file for Profile 21 with the prefix HBC4 will be named `HBC4-Profile21.txt`.
4. Enter a *Step Size* value so position and depth data is extracted at regular increments along the profile.
5. Select the *Ground Coordinates*:
   - *X, Y* (values are derived from the reference coordinate system in the field sheet)
   - *Distance* (from the start of the profile), Z
6. Click **OK**.

The profile data is exported to a text file in the selected directory.

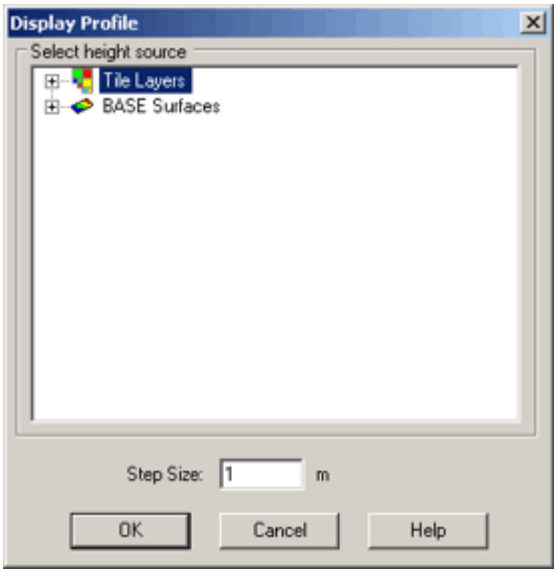
Display Profile

Display an existing profile in the Worksheet window.

1. Make sure that the Worksheet window is open.
2. Select a profile layer in the Control window.
3. Select the command.

Menu	Tools > Display Profile
Pop-up	Display (Control window)

The Display Profile dialog box is shown.



4. Select a height source (tile layers or BASE surface) by clicking the + icon and selecting the layer you want to use.
5. Type a *Step Size* to extract position and depth data from the profile line.
6. Click **OK**.

The profile is loaded into the Profile tab of the Display window.

Selected Soundings

A selection of soundings for display can be created in the Field Sheet Editor and opened in HIPS and SIPS as a sub-layer of the field sheet. The sounding selection is generated from a tile, grid or BASE Surface attribute layer. The selection mechanism is shoal- or deep-biased and allows control over the final density of the selected sounding layer. The sounding layer can be written to a CARIS file or to a Hydrographic Object Binary (HOB) file.

You can colour code the selected soundings according to depth, if necessary. See "SOUNDING COLOUR OPTIONS" ON PAGE 371.

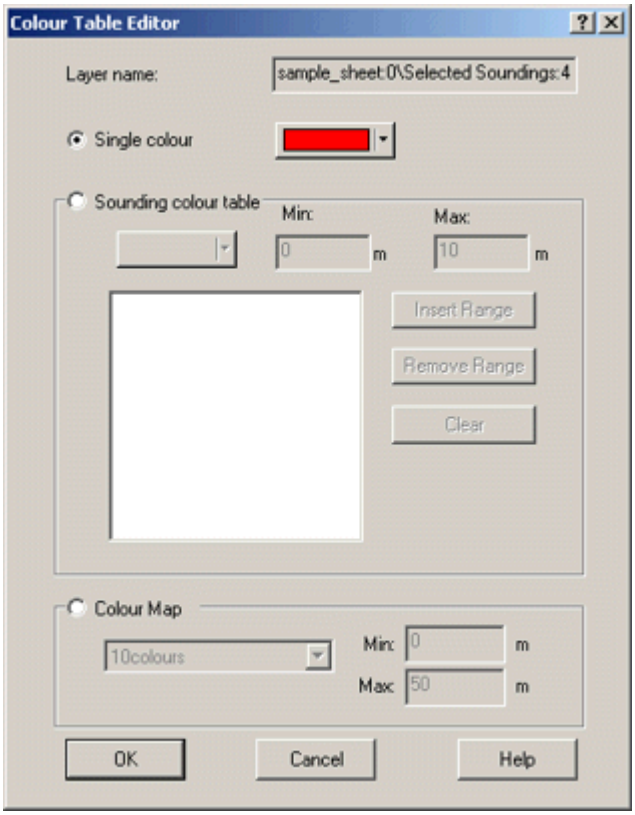
Sounding Colour Options

The Colour Table Editor lets you colour code soundings for easier display and interpretation.

1. Select a sounding layer in the Control window.
2. Select the Sounding Colours command.

Pop-up: Sounding Colours

The Colour Table Editor dialog box is displayed.



Single colour

Use this option to display a soundings in a single colour.

1. Select the *Single Colour* option.
2. Select a colour from the colour picker, or create a custom colour from the standard Windows colour palette.
3. Click **OK** to save and apply settings.

Sounding colour table

1. Select the *Sounding Colour Table* option.
2. Select a colour from the colour palette, or create a custom colour from the standard Windows colour picker.
3. Type a minimum range depth in the field beside the palette.
4. Type a maximum range depth in the next field
5. Click **Insert Range**.

The colour and range values are displayed in the Colour Table dialog box.

6. Repeat Steps 2 to 5 to continue adding colours.
7. To remove a single colour range, select a range so it is highlighted and click **Remove Range**.
8. To remove all selected ranges, click **Clear**.
9. Click **OK** to save and apply settings.

The soundings are colour coded according to the range values you selected earlier.

Standard colour map

Use this option to select a standard colour map.

1. Select the *Colour Map* option.
2. Select a colour map from the drop-down menu.
3. Type the minimum and maximum range values for the colour map.
4. Click **OK** to save and apply settings.

The layer objects are displayed according to the colour map settings.

16

Subset Editor

Overview

The Subset Editor is a tool for viewing and editing georeferenced sounding and lidar data (see "LIDAR DATA" ON PAGE 469). The Subset Editor differs from the Swath Editor in that soundings no longer only have an across-track and along-track relationship—but are now corrected for navigation, vessel heading, and other auxiliary sensors, so that each sounding has a latitude and longitude value attached to it.

Some of the feature of the Subset Editor are as follows:

- view data in a three-dimensional mode so features such as depressions and rises are visible.
- rotate data to view it from different angles
- flag soundings as needed:
  - rejected
  - accepted
  - outstanding
  - examined
  - designated
- query data
- view BASE Surface areas in the 3-D window
- create and modify subset tiles to track the areas that have been completed.
- view and edit CUBE Surfaces

To find out about creating Designated Soundings in the Subset Editor, see "DESIGNATED SOUNDINGS" ON PAGE 223.

Workflow

The following workflow is a suggested method for cleaning data with the Subset Editor.

The Subset Tiles options (Steps 1 and 2) can be excluded, and the Subset Editor can be directly opened into the track lines. However, if you are processing large areas of data, the subset tiling method is a useful way to track changes.

1. Make sure a field sheet (with BASE surface data) is open.

   (See "CREATE A FIELD SHEET" ON PAGE 349)

2. Create a Subset Tile in the field sheet area.

   (See "SUBSET TILES" ON PAGE 376)

3. Select the Subset Editor command and create a subset area with the subset bounding box in the tiled area.

   (See "SUBSET EDITOR INTERFACE" ON PAGE 380 and "SIZE AND RESIZE SUBSET" ON PAGE 387)

4. Use the 3D View Controls to adjust the view of the data.

   (See "THE 3-D VIEW" ON PAGE 389, "THE 2-D VIEW" ON PAGE 393)

5. Clean subset data.

   (See "SUBSET CLEANING" ON PAGE 398)

6. Classify data in subset area.

   (See "SUBSET TILE TRACKING" ON PAGE 379)

Subset Tiles

Subset Tiles are used to track areas within the project that have been examined and completely cleaned using the Subset Editor.

Each tile within a Subset Tile layer has one of three settings: Incomplete, Partially Complete, or Completed (see"SUBSET TILE TRACKING" ON PAGE 379).

The extents of the subset tile area can be determined in two ways:

- The tiles cover the entire field sheet area.
- The tiles cover areas of the field sheet that contain a height source (weighted grid or BASE Surface).

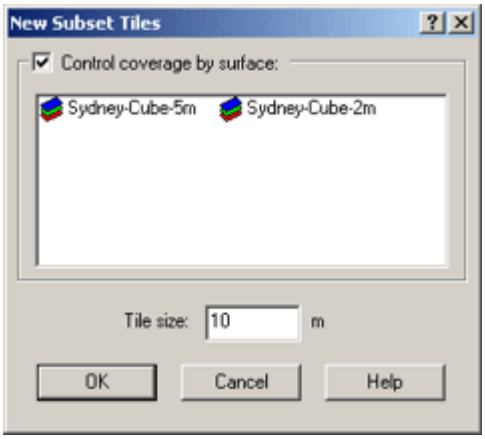
The option to cover the entire field sheet area is useful when you are surveying an area while also cleaning newly acquired data with the Subset Editor. You do not have to create a new subset tile layer because of additional data in your project.

Create a subset tile

1. Select the field sheet in the Layers tab of the Control window.
2. Select the New Subset Tiles command.

Menu	Tools > New Subset Tiles
Pop-up	New Subset Tiles

The New Subset Tiles dialog box is displayed.



The *Control Coverage by Surface* option constrains the subset tile area to a BASE Surface or weighted grid. When the check box is cleared, the subset tiles cover the entire field sheet area.

3. Select the *Control Coverage by Surface* check box to apply the subset tiles to a BASE Surface or weighted grid (If the check box is cleared, the section of the dialog box for selecting a height source is dimmed).
4. If you choose the *Control Coverage by Surface* option, select the BASE surface to be used as a template for a subset tile.
5. Type the tile size.
6. Click **OK**.

A Subset Tile layer is created and displayed in the field sheet area.

The Subset Tile layer name is added to the Session tab. The names are automatically generated beginning with SubsetTiles1. You can show or hide the layer with the check box beside the Subset Tile file name in the Session tab.

View subset data in a tile

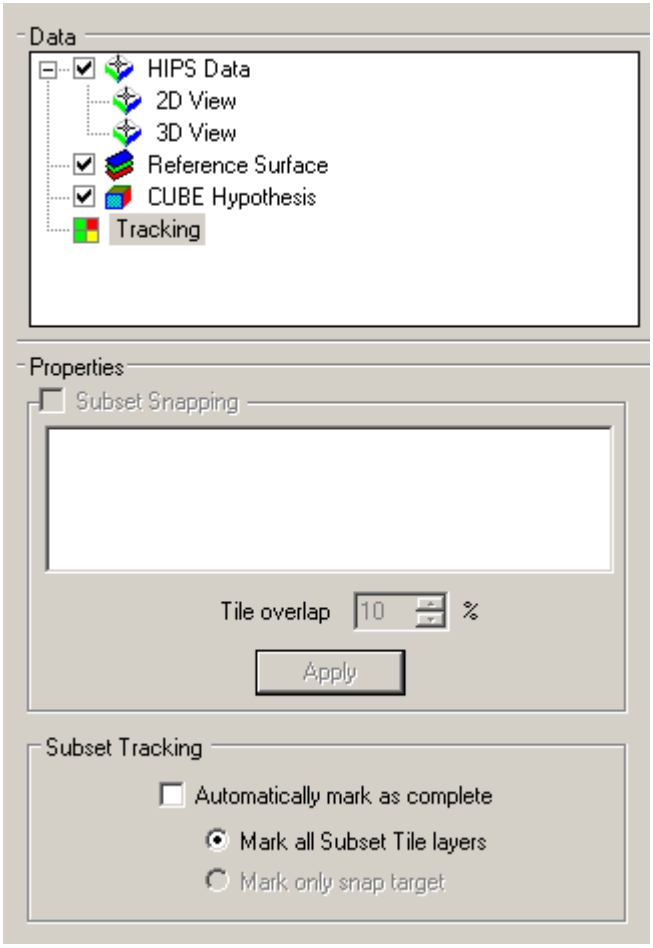
Subset data can examined and edited by selecting the Subset Editor command and dragging a box to define the area (see "Open Subset Editor" on page 380). You can clean data from the open subset (see "Subset Cleaning" on page 398).

Subset Tile Properties

Set subset tile properties for Subset Snapping and Subset Tracking. The properties are found in the Subset Editor tab of the Control window (see "Subset Editor Tab" on page 381), which is displayed when the Subset Editor is opened.

1. Make sure the Subset Editor tab is displayed.
2. Select the Tracking layer.

The bottom of the tab is refreshed to display the property options.



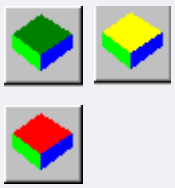
3. Select the Subset Snapping check box to activate the snapping options.
4. Select the *Tile Overlap* check box so that the subset bounding box overlaps the tile by a selected percentage. This option is useful when you have data that overlaps the tile area.
5. Click **Apply**.
6. Select the *Automatically Mark as Complete* check box to flag the data as examined and cleaned. This can be changed later (see "SUBSET TILE TRACKING" ON PAGE 379).

When the *Automatically Mark as Complete* check box is selected, two other options are made active.

7. Select the *Mark all Subset Tile layers* option to flag the tiles in ALL tile layers as completed, or select *Mark only snap target* to select tiles only in the selected tile layer as completed.

Subset Tile Tracking

Once you have examined and cleaned a subset, the area can be assigned one of three classifications.

Menu	Tools > Subset Editor > Complete/Partially Complete/Incomplete
Tool	

1. While the subset is still open, select a Cleaning Status command:
   - **Complete**: All data in the selected area is clean and ready for further processing or export.
   - **Partially Complete**: Not all the data within the area has been viewed and verified as clean in the Subset Editor.
   - **Reset**: Data has not been verified or cleaned.

The area is assigned the selected cleaning status. The area is shown in the Subset Tile as colour-coded according to the selected status classification:

- Complete = green
- Partially Complete = yellow
- Incomplete or not processed = red.

Only tiles totally encompassed by the subset bounding box are flagged with the selected status command. Tiles partially covered by the bounding box are flagged as Partially Complete (see “THE 3-D VIEW” ON PAGE 389).

Subset Editor Interface

Open Subset Editor

Track lines must be merged before the Subset Editor can be opened.

Menu	Tools > Subset Editor
Tool	
Pop-up	Tools > Subset Editor

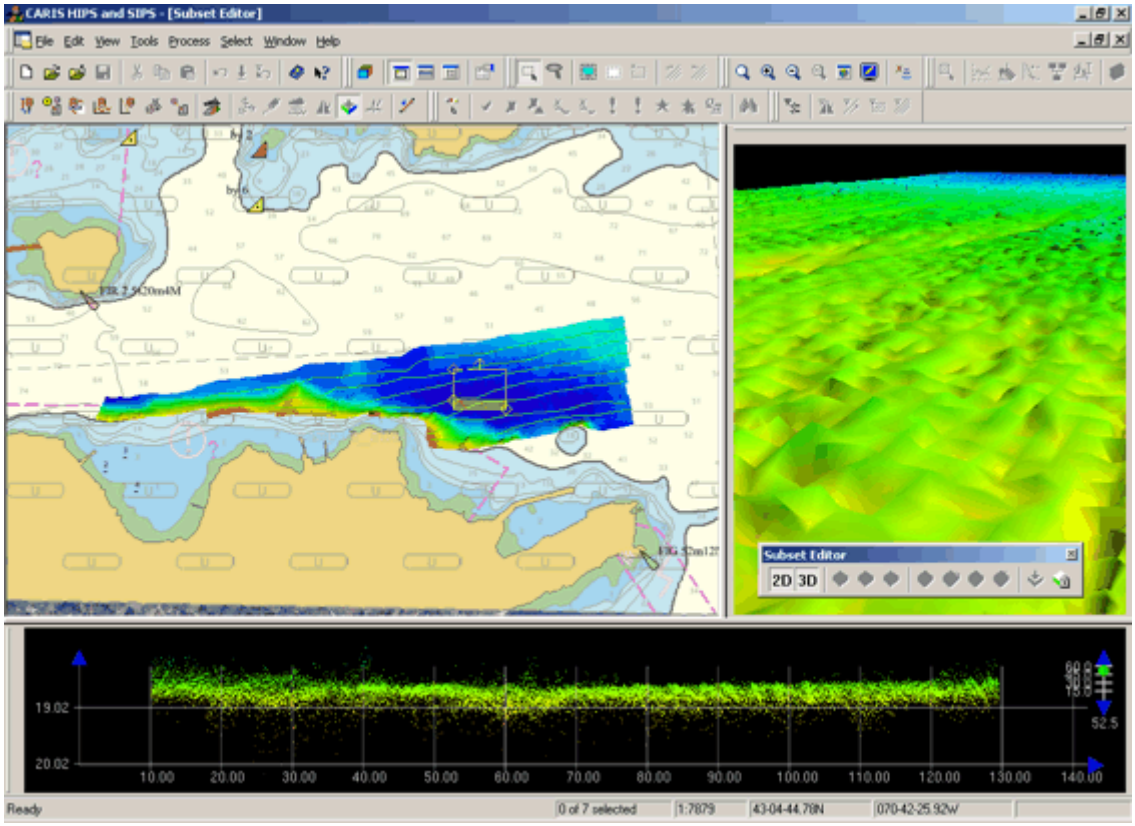
1. Select the Subset Editor command.
2. Press and hold the mouse button, and drag the cursor across the area of the Display window where you want to create the subset.

As the cursor is dragged across the Display window, a bounding box is drawn to show the subset area.

The subset is generated and displayed in the Subset Editor windows.

Menu	Tools > Subset Editor > Load Subset
Tool	

3. If no subset is displayed, select the Load Subset command to Below is an example of HIPS and SIPS with the Subset Editor.view the data in the Subset Editor windows.

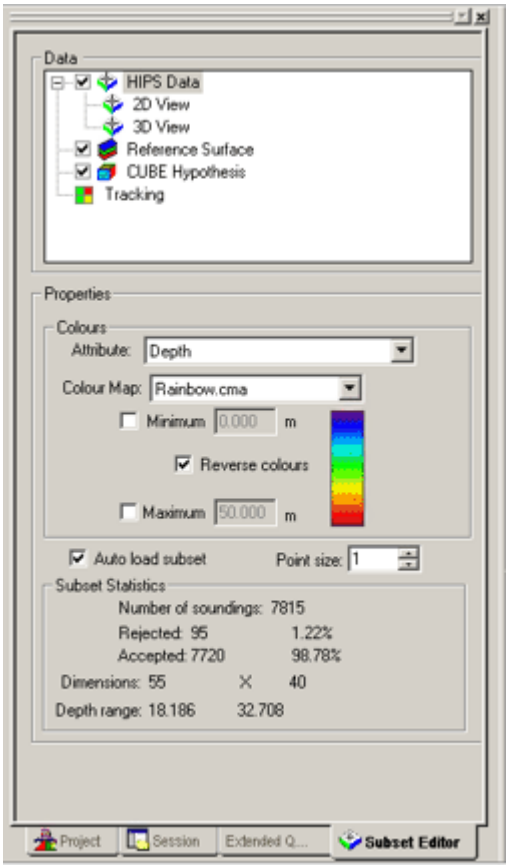


The Subset Editor has the following components:

- **Subset bounding box**: The bounding box selects an area where the subset is generated. Each time the box is moved or resized, a new subset is generated. The 2D slice box shows the area in the 2D window.
- **The 3D View**: The 3D (three-dimensional) View shows all soundings in the subset area. The view in the window can be rotated or changed.
- **The 2D View**: The 2D (two-dimensional) View displays soundings from the subset area that are within the 2D slicing box.

Subset Editor Tab

The Subset Editor tab is opened in the Control window when the Editor is started. The tab controls the data properties in the Editor. The tab is shown below.



The top part of the tab are the layers representing the data types. When you select a layer, the bottom part of the tab is refreshed to display the controls for the data properties.

The check boxes beside the layers control the display of data in the 3D and 2D Views. When the check boxes are selected and the layers are highlighted, data associated with that layer are shown in the 3D Display; when the check boxes are cleared, the data is no longer displayed.

The following layers are listed in the Subset Editor tab:

- **HIPS Data**: This layer controls the properties for sounding selection and display. The 3D and 2D sublayers control the display in their respective views.

  For more information on the controls related to these layers, see the following sections:

  - "View Subset Information" on page 384.
  - "Colour Options" on page 386
  - "The 3-D View" on page 389
  - "The 2-D View" on page 393
- **Reference Surface**: This layer controls the properties related to the display of BASE Surfaces in the 3D View. For

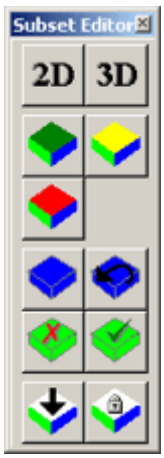
more information related to this layer, see the following section:

- "SUBSETS AND BASE SURFACES" ON PAGE 397.

- **CUBE Hypothesis**: This layer controls the display and editing properties related to CUBE data. For more information related to this layer, see the following section:
  - "HYPOTHESIS EDITING" ON PAGE 449
- **Tracking**: This layer controls the properties for subset tiles. For more information on the controls related to this layer, see the following section:
  - "SUBSET TILES" ON PAGE 376.

Toolbar

Commands related to the Subset Editor can be accessed from the Subset Editor toolbar. This is a floating toolbar that is automatically opened when the editor is started. It can moved and docked like other toolbars (see "MODIFY EXISTING TOOLBARS" ON PAGE 25).



View Subset Information

Each time a subset is generated, the following information is shown in the General layer of the Subset Editor tab:

- total soundings in the current subset
- total memory used by subset
- total accepted soundings
- total rejected soundings
- width
- height
- minimum depth level
- maximum depth level

Measure Distance

You can use the Measure Distance tool in the 2-D and 3-D Views to evaluate data problems, such as timing offsets, heave problems, etc.

Menu	Tools > Measure Distance
Tool	

1. Select the Measure Distance command.

A ruler icon is displayed next to the cursor.

2. Press and hold the mouse button and drag the cursor along the 2-D View or 3-D View. To maintain a straight horizontal line, press and hold the <Shift> key while drawing the line. To maintain a straight vertical line, press and hold the <Ctrl> key while drawing the line.

As the cursor is dragged across the View, a line is drawn from the start point to the point where the mouse button is released. The distance between the two points is shown along the line.

Colour Options

You can control how soundings are displayed in the subset.

1. Make sure the Control window is open.
2. Select the General layer in the Subset Editor tab.

The tab is refreshed to show the options related to soundings.

3. Select one of the following attributes from the drop-down list to colour code soundings:
   - **Depth**: The recorded depth level of the soundings.
   - **Amplitude**: Backscatter intensity values.
   - **Depth Uncertainty**: The vertical uncertainty values accorded to soundings.
   - **Position Uncertainty**: The horizontal uncertainty values accorded to soundings.
   - **IHO S-44**: The S-44 Survey Order ranking applied to soundings.
   - **IHO S-57**: The S-57 Zones of Confidence (CATZOC) attributes ranking applied to soundings.
   - **Project**: The project where the data was saved.
   - **Vessel**: The vessel(s) used on the survey.
   - **Day**: The day the sounding data was recorded.
   - **Line**: The track line in the project.
   - **Amplitude**: The sonar's wave pattern.
   - **Detection Method**: The phase detection method.

Colour Map

You can represent a range of values using a colour map. The colour ranges can be applied to other subsets so you have a standard colour code for displaying data.

- The colour at the top end of the selected map represents the minimum value (and all values less than the minimum value).
- The colour at the bottom of the map represents maximum value (and all values greater than the maximum value).
- The colours in between the minimum and maximum colours represent the depths and uncertainties between the defined range.

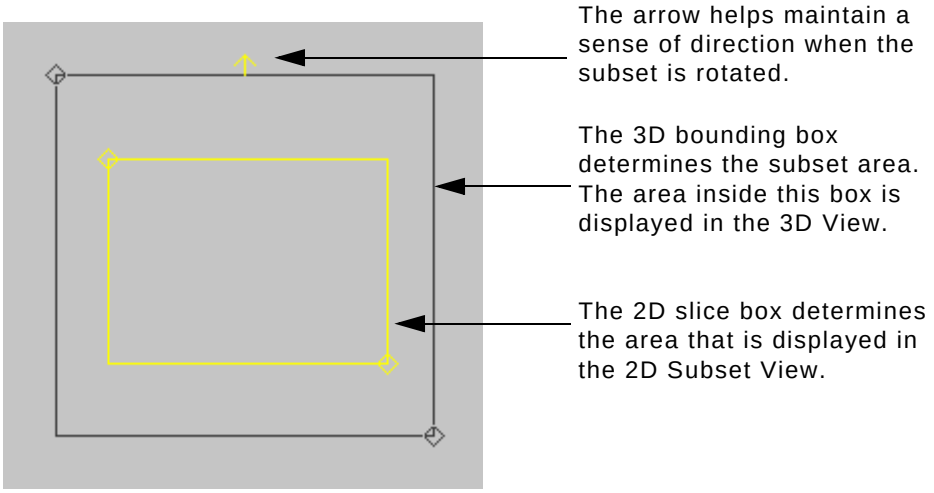
4. To set a minimum and maximum values, select these two check boxes and type the values in the respective fields. If these values are not set, then they are automatically read from the subset.
5. [Optional] Select the *Reverse Colours* check box to reverse the order in the colour map.

The display in the Views is refreshed to show the subset in the selected colours.

Size and Resize Subset

The size of the subset is determined by the bounding box. As the bounding box area is changed, a new subset is generated and the displays in the 2-D and 3-D Views are refreshed to show the new area.

Below is an example of the subset bounding box. Note that the shape has been altered for display purposes and does not match the box shape as it is first shown in the interface.



To resize the subset areas:

1. Position the cursor over any of the grips on the side of the boxes.
2. Press and hold the mouse button and drag the grip.

As the grip moves with the cursor, the box is resized to a new area.

You can also move the 2-D Subset slice up or down in the bounding box by pressing the arrow keys. The arrow at the top of the box shows the direction of the 2-D slice when the up arrow key is pressed.

When the box is resized, a new subset is generated.

To rotate the subset:

1. Press and hold the <Ctrl> key.
2. Position the cursor over any of the grips on the side of the box.
3. Press and hold the mouse button to rotate the bounding box to the desired angle.

Auto load subset

The *Auto Load Subset* option automatically loads subset data in the 3-D and 2-D Views when a subset area is created or changed. To create or adjust a subset area without immediately loading data, turn this option off.

1. In the General layer of the Subset Editor tab, select or clear the *Auto Load Subset* check box to turn this option to on or off (the default status is selected).
2. If you de-selected the *Auto Load Subset* option and want to load data manually, draw the subset and select the Load Subset command.

Data is loaded to the subset.

Menu	Tools > Subset Editor > Load Subset
Tool	

The Lock option locks the subset area so you do not accidentally resize or move the subset while moving it in the Display window. A new subset cannot be created while the lock is active.

1. Select the Lock command after creating a new subset or resizing a existing one.

The subset area is now locked into place and cannot be adjusted until the lock is removed.

Menu	Tools > Subset Editor > Lock
Tool	
Key	<L>

The 3-D View

The 3-D View displays the entire subset. You can use this view to examine and clean data.

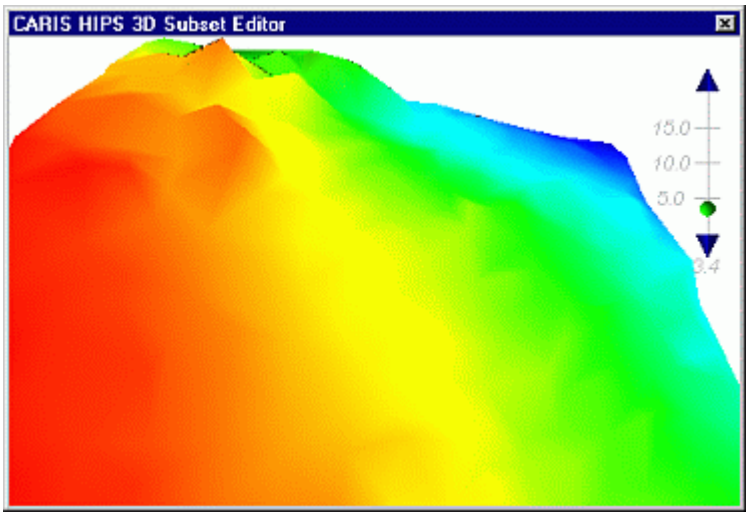
Open/Close the 3D View

Menu	Tools > Subset Editor > 3D View
Tool	3D

1. Select the 3D View command.

The view is displayed or removed from the HIPS interface.

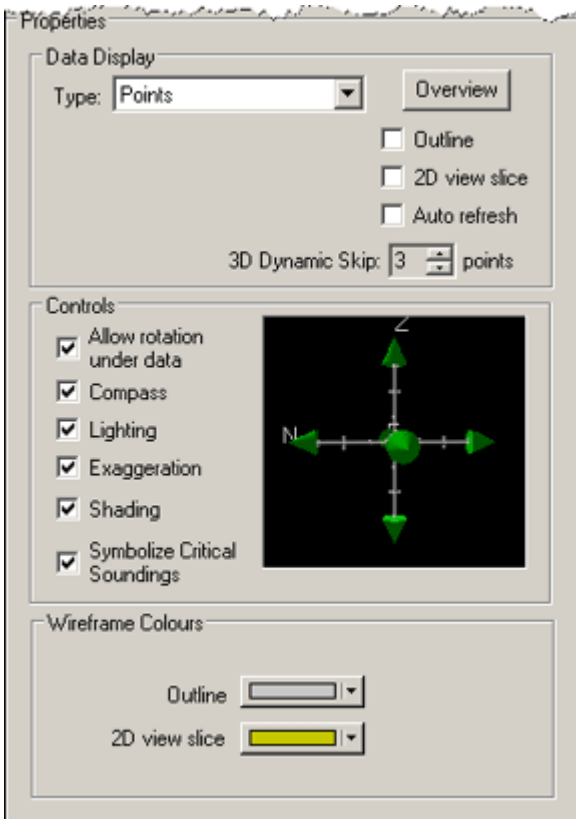
Below is an example of the 3D View.



All controls for the 3D View are contained in the 3D View sub-layer of the Subset Editor tab.

1. Click the Expand (+) icon beside the HIPS Data layer and select the 3D sub-layer in the Subset Editor tab.

The properties for the 3D View are displayed.



3-D Subset Display

Soundings are shown in the 3-D View according to the display type list at the top of the 3-D tab.

1. Select the display type from the list.

When the display type is selected, the 3-D View is refreshed to show the soundings according to the display type. The options for display type are shown in the tab when the display type is selected.

The options for each display type is explained in the following table.

Display Type	Definition	Options
Surface	The soundings are shown as a smooth surface.	Soundings are drawn to the 3-D View either by selecting the shoalest sounding in a user-defined area (or bin), or by selecting all soundings. 1. Select the *Shoal Bin Size* to select the shoalest sounding in an area (bin), or select *Use All Points* to display every sounding. 2. If you selected the *Shoal Bin Size* option, type the size of the area for selecting the shoalest sounding. 3. Click **Regenerate**.
Surface Wireframe	The soundings are shown as a Triangulated Irregular Network (TIN) that connects soundings.	

Display Type	Definition	Options
Points	Soundings are shown as unconnected points.	3D Dynamic Skip option determines how many sounding profiles and beams are displayed. For example, the value 6 means that only every 6th profile and 6th beam are shown in the view. 1. Select a skip value by clicking the up or down arrow buttons, or directly typing a value in the *3-D Dynamic Skip* field. 2. Click **Regenerate**.
Spheres	Soundings are shown as unconnected spheres.	
Cylinders	Soundings are shown as cylinders. If the soundings contain horizontal and vertical uncertainty values, then the cylinders are sized according to those values. For example, soundings on the outer edge of a swath have a larger radius than soundings on the inner edge because of the greater horizontal uncertainty. Soundings without TPE values are drawn to a set size.	

Rotation, illumination and exaggeration options

The compass controls rotation, the sun dial controls illumination, and the vertical exaggeration slider controls exaggeration.

2. Select or clear the *Compass*, *Lighting* and *Vertical Exaggeration* check boxes to show or hide these controls in the 3D View.

Pan and zoom

The pan, zoom and rotation functions are the same as in the 3-D View in the Swath Editor. See "PAN AND ZOOM" ON PAGE 316 for more information.

Shading

Darken the image in the 3-D View.

3. Select the *Shading* check box.

Symbolize critical soundings

Display soundings with Outstanding, Critical and Designated flags.

4. Select the Symbolize Critical Soundings check box.

Overview

Reset the View so the entire subset area is visible (default).

5. Click **Overview**.

Show subset outline

Display an outline of the subset bounding box.

6. Select the *Show Subset Outline* option. When this option is checked, the outline is displayed in the 3D View.

Show 2D slice

Display an outline of the 2D slice area.

7. Select the *Show 2D* slice option. When this option is checked, the 2D slice outline is displayed in the 3D View.

Allow rotation under data

This option rotates the 3D display by 180° to view soundings underneath the subset. When this option is not selected, the subset can only be rotated by 90°.

8. Select the *Allow Rotation Under Data* option to rotate the subset display by 180°.

Centre on sounding

Recentre the display on a sounding.

9. Select a sounding in 3-D View by clicking on it with the middle-mouse button.

The display in the 3-D View is re-centred to the selected sounding.

Wireframe colours

Select a colour to display 3-D and 2-D outline in the 3-D View.

10. Select a colour from the picker, or click **Other** and create a custom colour from the palette.

The 2-D View

The 2D View displays a profile of soundings within the 2D slicing box. You can use this view to examine and clean data.

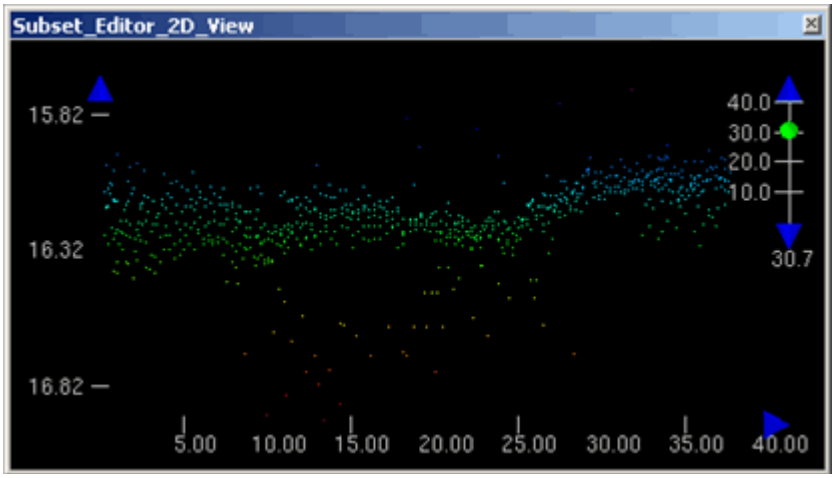
The location and size of the cutaway view is determined by the size of the 2D slice box (see "SIZE AND RESIZE SUBSET" ON PAGE 387). As the slice box is re-sized or moved to another position in the subset, the display in the View changes accordingly.

Open/Close the 2D View

Menu	Tools > Subset Editor > 2D View
Tool	

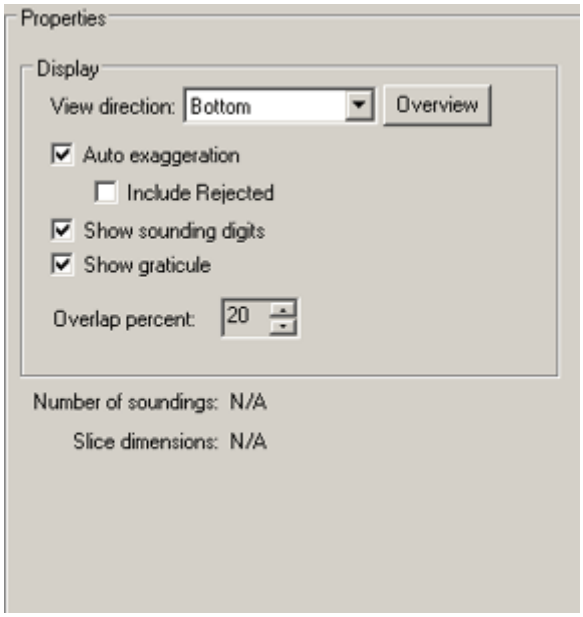
1. Select the 2D View command.

Below is an example of the 2D View.



All controls for the 2D View are in the 2D View sub-layer in the Subset Editor tab.

2. Click the Expand (+) icon beside the General layer and select the 2D sublayer in the Subset Editor tab



The 2D View tab contains the following information:

- number of soundings in the 2D slice
- dimensions of the slice (height x width)

When the 2D slice bounding box is resized, the information in the tab is refreshed with new data.

Change view perspective

When the 2D View is first displayed, the view perspective is determined by the dimensions of the 2D Slice Box.

- If the height of the slice box is greater than the width, the view is automatically shown from the left side.
- If the width of the slice box is greater than the height, the view is automatically shown from the bottom.

3. Select any of the following options from the *View Perspective* drop-down list to change the view angle:
   - top (view data from the front of the slice box)
   - right (view data from the right side of the slice box)
   - centre (view data from above the slice box)
   - bottom (view data from the back of the slice box)
   - left (view data from the left side of the slice box)

The 2D View is refreshed to show the data from selected angle.

4. To return to the default view, select Automatic from the drop-down list.

2D vertical exaggeration

The vertical exaggeration is the ratio of the vertical scale versus the horizontal scale. You can set the vertical exaggeration scale with the vertical exaggeration scale in the 2D View.

1. Position the cursor over the ball on the vertical exaggeration scale.

2. Press and hold the mouse button, and move the ball up or down the scale.
3. To change scale values, click the arrows at either end of the vertical exaggeration scale in increase or decrease the scale by a value of 10.

As you move the ball, the data in the 2D View is rescaled.

The *Auto Exaggeration* option determines the scale automatically by using all visible depths.

When *Auto Exaggeration* mode is on, the *Include Rejected* option controls whether or not the rejected data is used to determine the minimum and maximum sounding depth to be displayed in the 2D View.

4. Select the *Auto Exaggeration* check box.
5. Select the *Include Rejected* option to resize the display in the view to include rejected data.

The 2D View is adjusted to display data according to the maximum and minimum settings.

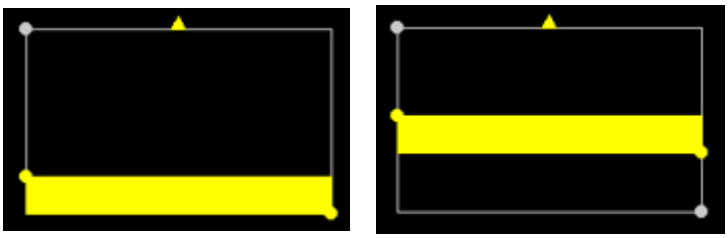
Display options

The display can also be modified to draw sounding values in the View and display a grid with depth and distance values.

1. Select the *Show Sounding Digits* check box to draw sounding values in the View.
2. Select the *Show Graticule* check box to display a grid representing depth and distance. If the Show Graticule check box is cleared, the grid is removed from the 2-D View, but the values representing distance and depth are still displayed. The vertical scale represents depth and the horizontal scale represents distance. The values shown in the 2D View are determined by the display option:
   - If the top/bottom/left/right display options are selected, both the depth and distance are shown.
   - If the centre display option is selected, distance in both directions are displayed.

Move 2-D Display

You can move the 2-D Slice box within the subset by pressing the arrow keys.



The 2-D Slice Box is moved up as the up arrow key is pressed.

1. Press the arrow keys to move the 2-D Slice Box within the subset.

2. Select an *Overlap Percent* to determine how much of the 2-D Slice Box overlaps the previous area in the 2-D View when the box is moved.

Pan and zoom options

The pan and zoom functions are the same as in the 3D View in the Swath Editor. See "PAN AND ZOOM" ON PAGE 316 for more information.

You can also pan in the Subset Editor Views by clicking on a sounding with the middle-mouse button. This will shift the subset so that the sounding is centred in the View.

To return the display in the View to original setting before zoom and pan operations, click **Overview**.

Close Subset Editor

Menu	Tools > Subset Editor
Tool	
Pop-up	Tools > Subset Editor

1. Select the Subset Editor command.
2. If changes were made to the data, you are prompted to save them.

When data in the Subset Editor is changed and saved, the ProcessedDepths and ObservedDepths files are updated to reflect the changes.

3. Click **OK** to save changes.

The Subset Editor is closed and no longer visible in the HIPS interface.

Subsets and BASE Surfaces

BASE Surfaces can be shown in the 3-D View. This feature is used for the following quality assurance measures:

- Determine the position of Designated soundings against an existing BASE Surfaces.
- Determine if Designated soundings have been correctly applied to a Finalized BASE Surface.

A BASE Surface is displayed by depth attribute.

1. Select the *Reference Surface* layer to view the properties.

The tab is refreshed to show the property options for the a BASE Surface layer.

2. Select the BASE Surface from the file tree.
3. Determine the height of the BASE Surface (relative to the subset) using an attribute by selecting the attribute from the *Height from Attribute* drop-down list.
4. Although the depth attribute is used to display the Surface, you can colour the Surface by other attributes by selecting the attribute from the *Colour by Attribute* drop-down list.
5. Display the surface as a points, wireframe or a complete surface by selecting an option from the *Display* drop-down list.
6. Select a transparency level for the Surface, by moving the up or down arrow buttons.
7. Click **Load**.

The BASE Surface is displayed in the 3-D View according to the above options.

8. [Optional] Click **Unload** to remove the BASE Surface from the 3-D View.

Subset Cleaning

Soundings in the 2D and 3D Views can be interactively examined and their status flags can be modified.

When selecting data in the 3D View, remember that all data in the highlighted area is selected, not just the visible data.

Reject

Each sounding in the HIPS format carries several status flags that the software turns on and off as needed. Each sounding begins as an Accepted sounding and if rejected, for any reason during processing, it can always be returned to the accepted status without loss of data.

A sounding rejected within the Subset Editor has the Rejected by Hydrographer status flag turned on.

1. Select the data you want to reject so it is highlighted.
2. Select the Reject command.

Menu	Edit > Status Flag > Reject
Tool	
Pop-up	Reject
Key	<R>

The sounding status flag, Rejected by Hydrographer, is now on and the sounding may or may not be displayed depending if the rejected data is toggled to visible.

Accept

Change rejected soundings back to the accepted status.

1. Select the rejected data that you want to accept.
2. Select the Accept command.

The soundings are now marked as accepted.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

Designated Soundings

The Designated Sounding flag identifies the shoalest sounding in a feature. The purpose of the Designated Sounding flag is to ensure that the shoalest depths over significant seabed features are maintained in charts and other standard hydrographic products. For example, in a cluster of soundings surrounding an outcrop of rocks, the shoalest sounding among that cluster is identified as Designated.

For more information, see "DESIGNATED SOUNDINGS" ON PAGE 223.

1. Select the shoalest sounding from a cluster of soundings around a feature in the Subset Editor.
2. Select the Designated Soundings command.

The sounding is flagged as Designated.

Menu	Edit > Status Flag > Designated
Tool	
Key	<D>

Outstanding

Flag a sounding as requiring further examination.

1. Select the data that you want flag as Outstanding.
2. Select the Outstanding command.

Menu	Edit > Status Flag > Outstanding
Tool	
Pop-up	Outstanding

Examined

Flag questionable soundings as having been examined and verified.

1. Select the data that you want to flag as Examined.
2. Select the Examined command.

Menu	Edit > Status Flag > Examined
Tool	
Pop-up	Examined

Query

View information on selected data.

1. Select the data that you want to query.
2. Select the Query command.

The data is displayed in the Worksheet window.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

17

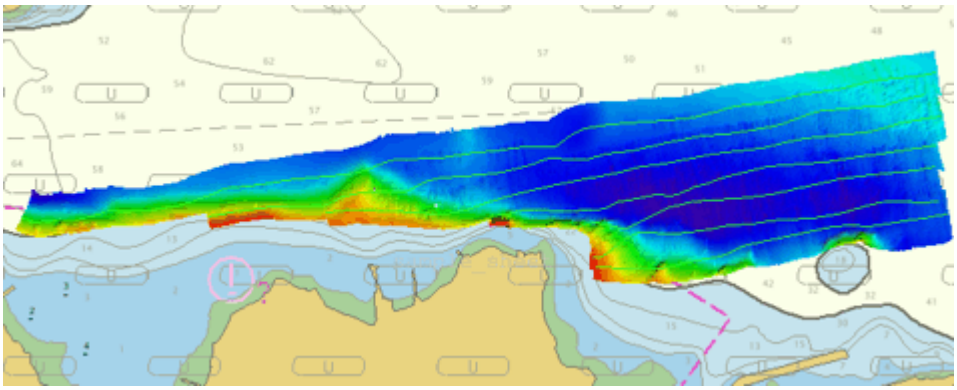
BASE Surfaces

Overview

The Bathymetry Associated with Statistical Error (BASE) Surface is a georeferenced image of a multi-attributed, weighted-mean surface. A BASE Surface can also contain a visual representation of horizontal and vertical uncertainty.

A BASE Surface file is saved with a HNS extension (HIPS Navigation Surface). An XML file with the same name as the BASE Surface is also generated when the Surface is created. This file contains the metadata for the Surface.

You can select specific track lines to include in the BASE Surface or let the wizard select them for you automatically. You can generate as many Surfaces as necessary within a field sheet. An example of a BASE Surface image is shown below



The BASE Surface meets the needs of various users. The primary focus of current survey products is chart production by creating a shoal-biased bin where shoals are exactly preserved but finer seafloor features are omitted. Other users—marine geologists, coastal zone management, fisheries habitat management, and ocean engineering—require a more detailed view of the seafloor. The BASE Surface can be generated to provide either safety of navigation or a detailed examination of the seafloor.

The BASE Surface is used for the following purposes:

- A background visual aid when cleaning data interactively and automatically.
- A georeferenced TIFF that can be exported to other software such as real-time navigation systems.
- Data can be exported as an XYZ text file and imported as soundings in a CARIS map or used to build digital terrain models in other software.
- A Product Surface that can be used in the Field Sheet Editor to build contours, sounding selections, profiles, and other data layers. This data can later be used to create an ENC or raster chart.
- A quality control feature for data. By viewing the uncertainty values for nodes, you can identify bad data and determine if an area needs to be re-surveyed.

All BASE Surfaces use range weighting to determine how a sounding is applied to a node. Range weighting is based on distance—soundings close to a node are given greater weight than soundings further away. The calculated node positions are determined by the corner coordinates of the field sheet.

Three optional surface types are used in creating BASE Surfaces:

- Uncertainty uses a weighting scheme based on depth uncertainty as specified by a selected S-44 survey order.
- Grazing angle uses a weighting scheme based on a beam's intersection angle with the seafloor.
- Combined Uncertainty and Bathymetry Estimator (CUBE) surface uses multiple hypotheses to represent potential depth variances along the seafloor (see "CUBE PROCESSING" ON PAGE 437).

Range Weight

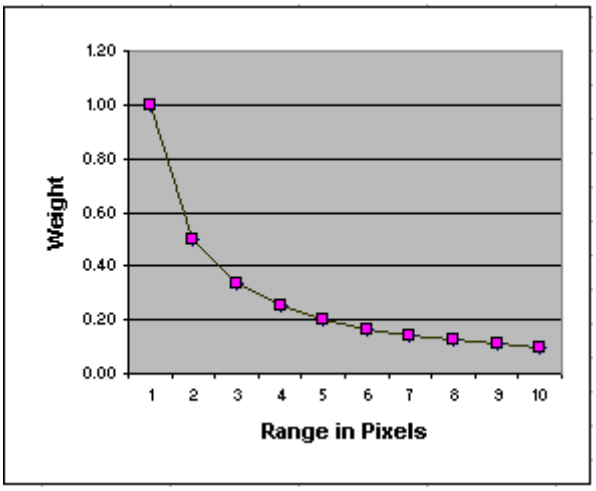
The range weight is inversely proportional to distance from the node: soundings closer to a node are given a greater weight than soundings further away.

The number of nodes each sounding is applied to is determined by the size of the beam footprint. The beam footprint is calculated using depth, sonar beam width, and the grazing angle.

The sonar that is used, with the appropriate beam width, is defined in the BASE Surface wizard.

A list of multibeam systems and corresponding beam widths is listed in ..\Hips\System\DeviceModels.xml. You can modify this file.

The following graph demonstrates range weighting using distance from a node in units of pixels (multiples of the BASE Surface resolution).



Uncertainty Weight

The weight that a sounding contributes to a node is inversely proportional to the predicted depth uncertainty of the sounding.

The sounding's depth uncertainty is scaled as a function of the sounding's distance from the node. The propagation of the depth uncertainty to the node takes into account the sounding's distance from the node AND the sounding's horizontal uncertainty. Thus, both depth uncertainty and horizontal uncertainty play a role in determining the weight a sounding contributes to a node.

The sounding's area of influence is a radius that determines the number of nodes to which the sounding can be applied. This radius is determined by the depth uncertainty of the selected IHO S-44 survey order (see formula below).

$$\textit{Depth Uncertainty} = \sqrt{a^2} + (b \times d)^2$$

Once the depth uncertainty is propagated beyond the selected S-44 survey order requirement for depth accuracy, then it has reached its area of influence limit.

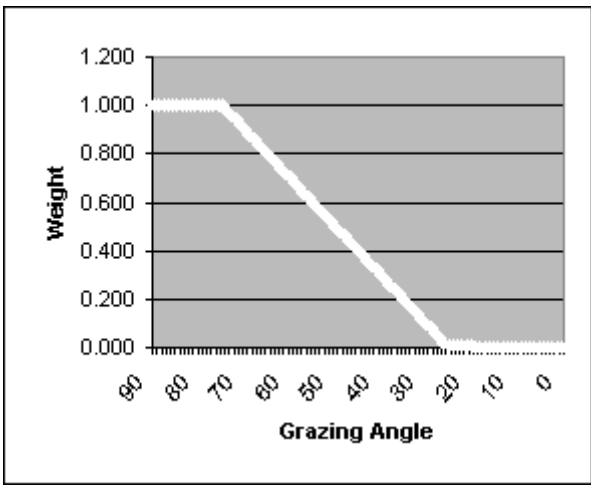
All distance computations are based on the ground coordinate system defined for the BASE Surface—they are not simplified to cell distances. All of the nodes within a sounding's area of influence are located rigorously. No simplified weight matrix scheme is used.

Swath Angle Weight

The weight a sounding contributes to the Surface also varies with the sounding's grazing angle with the seabed.

This weighting value is important in areas with adjacent or overlapping track lines. The swath angle weight ensures that higher weight is given to beams from the inner part of a swath than to outer beams from adjacent track lines.

In the following graph, beams with a grazing angle between 90 and 75 degrees are given a weight of 1.0. The weight decreases linearly to 0.01 as the angle with the sea floor decreases to 15 degrees. This is the default swath angle weight scheme as defined by the file ..\Hips\System\GrazingAngleWeights.txt. The default file can be customized to the weighting scheme you want.



BASE Surfaces and Weighted Grids

Weighted grids are mean-averaged seabed surfaces generated from processed bathymetry in previous versions of HIPS. While grids share many similarities with BASE Surfaces, there are fundamental differences between the two products:

- Horizontal and vertical uncertainty data are not available in weighted grids.
- Attribute values that can be displayed in BASE Surfaces are not available in weighted grids.
- Range Weighting and Swath Angle Weighting schemes were both applied when the grid was generated while Swath Angle Weighting is optional for BASE Surfaces.

You can open weighted grids in HIPS 6.1 and grids can be used to create tiles, selected soundings, contours and other data products in the Field Sheet Editor. Grids can also be exported to an ASCII file format or a georeferenced TIFF image. However, you cannot apply BASE Surface functionality (Finalize, Combine, Rebuild, etc.) to a grid.

> BASE Surface operations cannot be applied to weighted grids.

To build a BASE Surface with the same depth values as a weighted grid, you should select the field sheet that contains the grid and create a BASE Surface using the Swath Angle Weighting option (see "CREATE BASE SURFACE" ON PAGE 408).

Create BASE Surface

BASE Surfaces are created using a wizard. All track lines must be merged before a BASE Surface can be created.

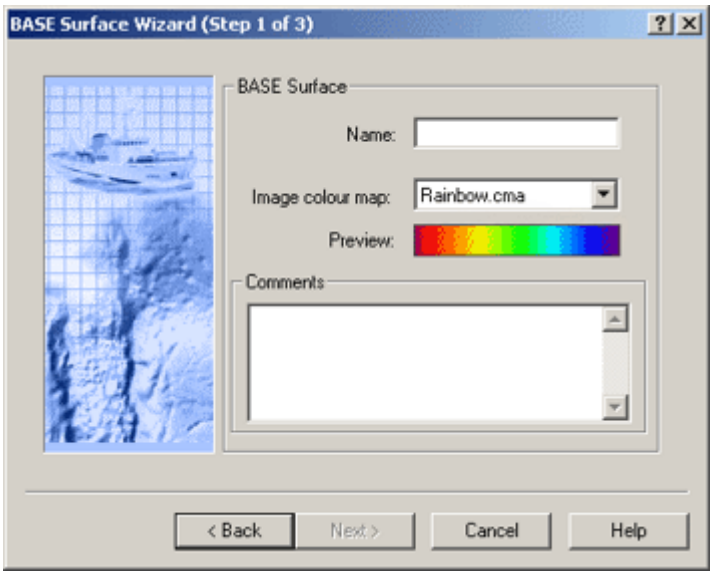
1. Select the area for the BASE Surface:
   - To create a BASE Surface for the entire field sheet area, select a field sheet in the session data tree in the Control window.
   - To create a BASE Surface for a single track line, select a track line in a field sheet and then select the field sheet layer in the data tree.
2. Select the New Base Surface command.

Menu	Process > BASE Surface > New
Tool	

The number of dialog boxes displayed in the wizard is determined by the type of surface you are creating.

BASE Surface (Step 1)

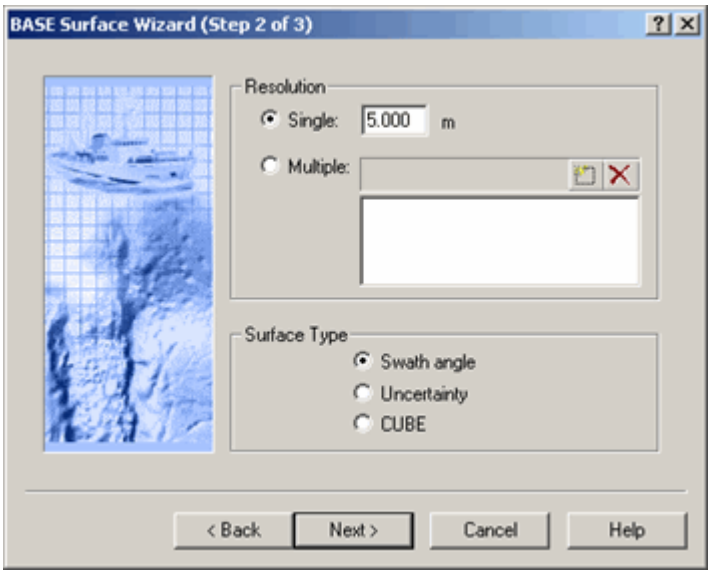
The BASE Surface Wizard (Step 1) dialog box opens.



1. Type a *Name* for the BASE Surface.
2. Select an *Image Colour Map* file for the TIFF image that is created after the Surface has been computed. You can change the colour map later.
3. Type any *Comments* (if necessary) in the text box.
4. Click **Next**.

BASE Surface (Step 2)

The BASE Surface Wizard (Step 2) dialog box is displayed.



The *Resolution* value(s) sets the distance(s) between BASE Surface nodes. You can use a single resolution for the entire surface, or, use multiple resolutions for a range of depths.

To enter a range of values, you must select the select the *Multiple* option. Values are entered in the following format:
minimum depth - maximum depth, resolution.
For example, 0.0-10.0,5

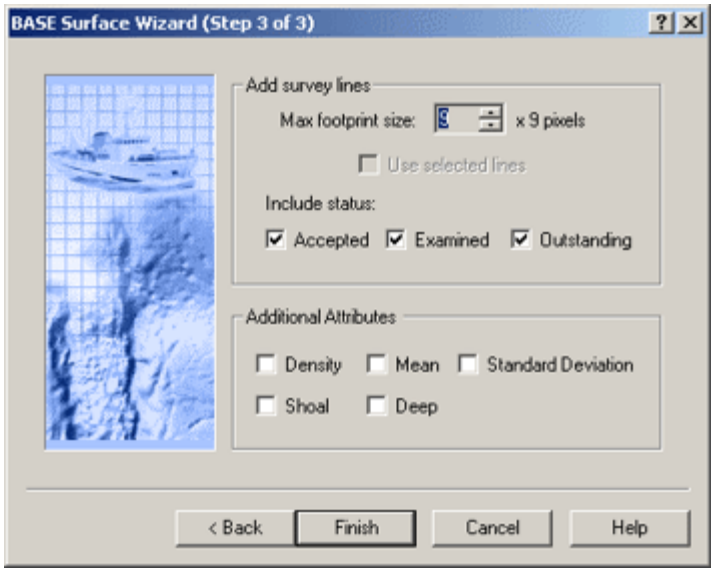
5. Select the *Single* field and enter a resolution value, or select the Multiple option and enter a range of values.
6. Select either *Uncertainty Weighting*, *Swath Angle* or *CUBE*. The next dialog box is determined by the weighting option you choose.
7. Click **Next**.

BASE Surface (Step 3)

The third dialog box is displayed.

Swath Angle Weighting

If you selected the *Swath Angle Weighting* option, the following BASE Surface Wizard (Step 3) dialog box is displayed.



The *Max Footprint Size* defines the maximum area in the Surface to which a sounding is applied. Use this value to prevent over-expansion of the footprint due to large depth spikes or very shallow grazing angles.

1. Click the up and down arrow buttons in the *Max Footprint Size*.

If one or more track lines were selected before the BASE Surface process was started, the *Use Selected Lines* check box is enabled.

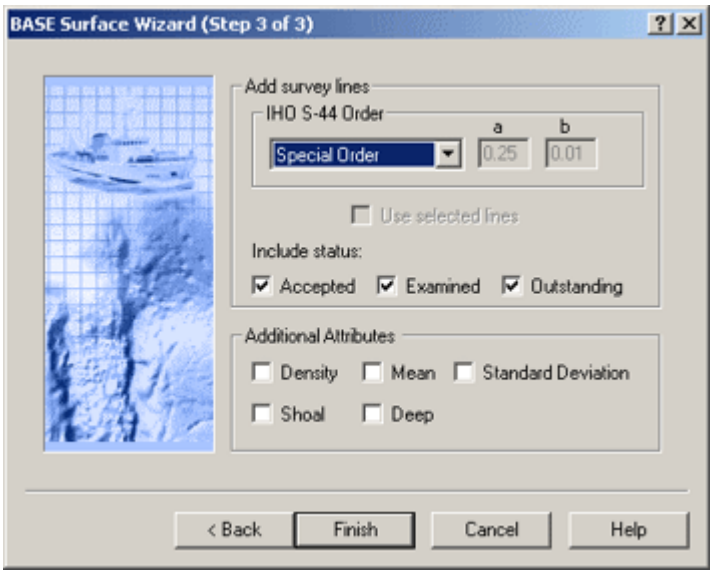
2. Clear this check box if you want apply a BASE Surface to the field sheet area.

The *Accepted* check box is selected by default.

3. Select either the *Examined* or *Outstanding* check boxes to include data with these status flags in the BASE Surface process.
4. Select *Additional Attributes* to be displayed in the BASE Surface.
5. Click **Finish**.

Uncertainty weighting

If you selected the *Uncertainty* option, the following BASE Surface Wizard (Step 3) dialog box is displayed.



1. Select an *IHO S-44 Order* from the drop-down list.

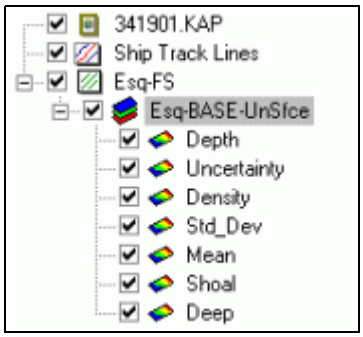
The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. The values are read from ..\HIPS\System\IHO_Standards.xml.

2. If one or more track lines were selected before the BASE Surface process was started, the *Use Selected Lines* check box is enabled. Clear this check box if you want to apply the BASE Surface to the field sheet area.
3. Select either the *Examined* or *Outstanding* check boxes to include data with these status flags in the BASE Surface process. The *Accepted* check box is selected by default.
4. Select the attributes to be displayed in the BASE Surface.
5. Click **Finish**.

The BASE Surface is created.

View BASE Surfaces

In the HIPS interface, A BASE Surface layer is created and displayed in the Session tab of the Control window. Attributes are displayed as sub-layers of the BASE Surface.



1. Make sure the Session tab of the Control window is open.
2. Click Expand (+) icon beside the Field Sheet file name to expand the directory tree.
3. Display the BASE Surface by selecting the check box beside the layer.

The following attributes can be displayed in the BASE Surface (depending on the weighting method you selected in Step 6 and the selected attributes in Step 7):

- **Depth**: Depth at the node.
- **Density**: The number of soundings contributing to a node.
- **Standard Deviation**: The number of standard deviations that the node is from the mean.
- **Uncertainty**: The depth uncertainty assigned to each node.
- **Mean**: The mean depth calculated from all soundings contributing to a node.
- **Shoal**: The shoalest sounding contributing to a node.
- **Deep**: The deepest sounding contributing to a node.

4. To view or hide layers, select or clear the check boxes beside the layer.

When an attribute layer is checked, a ToolTip containing the attribute's value at the node is displayed when running the cursor across the area of the Surface.

Properties

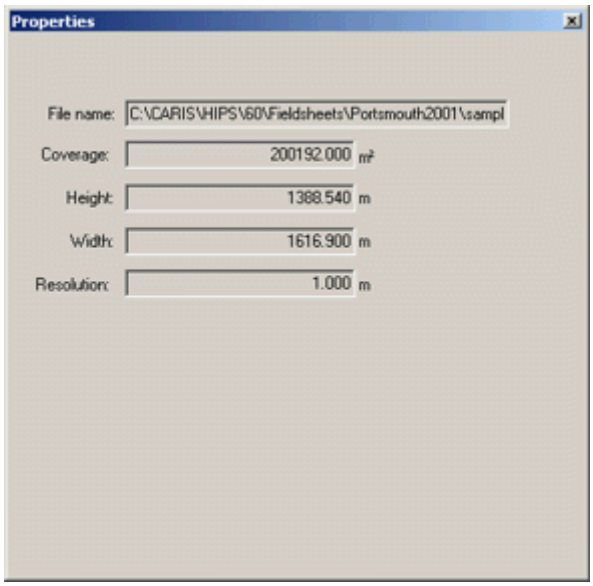
You can use the Properties window to view basic information about a BASE Surface or to set the display options for a BASE Surface attribute layer.

View Surface Properties

1. Select a BASE Surface layer in the Session tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up window.

Menu	Window > Properties
Tools	
Pop-up	Properties

The Properties window for a BASE Surface is displayed.



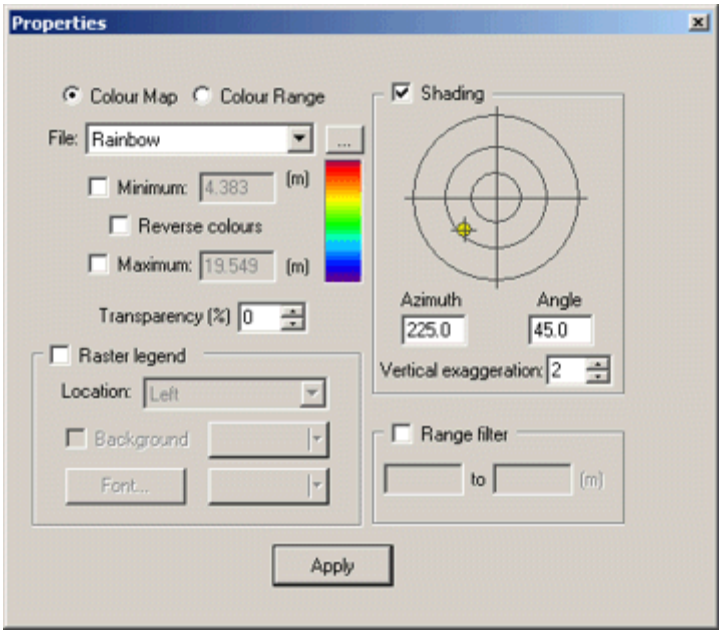
The *File Name*, *Resolution* and area fields displayed in the window. These are read-only and cannot be modified.

Set Attribute Properties

Menu	Window > Properties
Tools	
Pop-up	Properties

1. Select a BASE Surface attribute layer (depth, shoal, mean, etc.) in the Session tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up window.

The Properties window for the BASE Surface attributes layer is displayed.



3. Select any of the following colour map options:
   - *Colour Map*: A list of pre-defined colour maps available for images. These colour maps are located in the ..\Hips\System directory.
   - *Colour Range*: Custom colour ranges showing variances in attribute properties. Colour ranges can be saved as CRF files.

If you select *Colour Map*, the following controls are displayed:

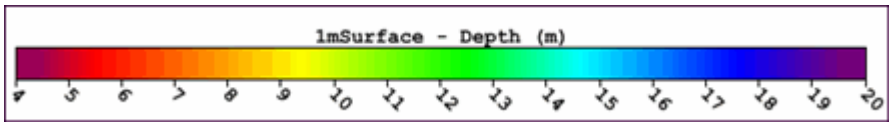
- *Minimum*: All attribute values equal to or less than this value are assigned the shallow end of the colour map.
- *Maximum*: All attribute values equal to or greater than this value are assigned the deep end of the colour map.
- *Reverse Colours*: Change the order in the colour map.
- *Transparency*: A percentage level for viewing other features through the Surface area.

If you select *Colour Range*, the window is refreshed to not display the controls associated with the colour map.

4. Select a colour map or colour range file from the *File* drop-down list.

5. If necessary, click Browse to modify existing colour maps or colour ranges, or create new ones (see "COLOUR MAP EDITOR" ON PAGE 417 and "COLOUR RANGE EDITOR" ON PAGE 419).

The *Raster Legend* option displays a scale of ranges in the selected colour map.



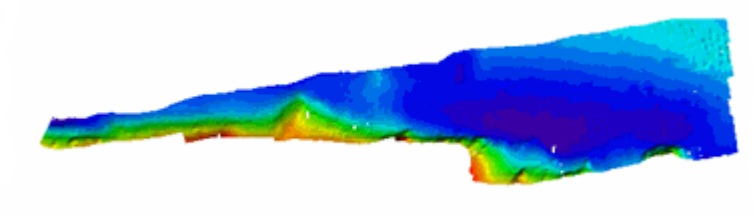
6. Select from any of the following legend options:
   - *Raster Legend*: Display a legend containing the colour map for an attribute and corresponding attribute values.
   - *Location*: Position of the legend in the Display window.
   - *Background*: Colour for the background area of the legend.
   - *Font*: The type, style and size of font used for text and values in the legend. These values are selected from the standard Windows Font dialog box that is displayed when the Font button is clicked.
7. Select *Background* and *Font* colours from the respective colour pickers, or create custom colours from the Windows colour palettes.

The *Shading* option controls the lighting angles of the BASE Surface. Using this option may highlight fine scale features of the Surface that would not normally be seen.

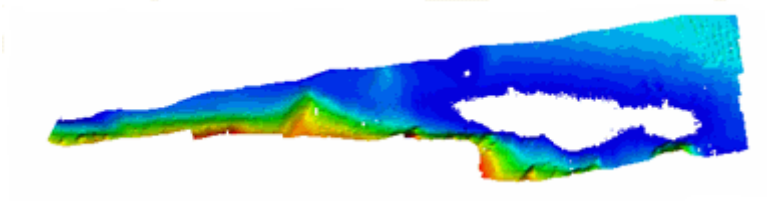
The sundial control sets the shading effect in the image. The *Elevation* value varies from 90 to 0 degrees starting from the middle of sundial. The *Azimuth* value varies from 0 to 360 degrees in a clockwise direction.

8. Select the *Shading* check box to activate the sundial and *Vertical Exaggeration* options.
9. Position the cursor over the button in the sundial and press and hold the mouse button to drag the button to a new position. The *Elevation* and *Azimuth* values change as the button is dragged.
10. Change the exaggeration of the illuminated sea floor by selecting a *Vertical Exaggeration* value. Increasing the exaggeration may improve the view of the fine scale features on the seabed.

The *Range Filter* option controls the value ranges that are displayed in the BASE Surface. An example of using the filter controls is displayed below.



BASE Surface before filtering. All depth values ranging from 0 metres to 24 metres are displayed.



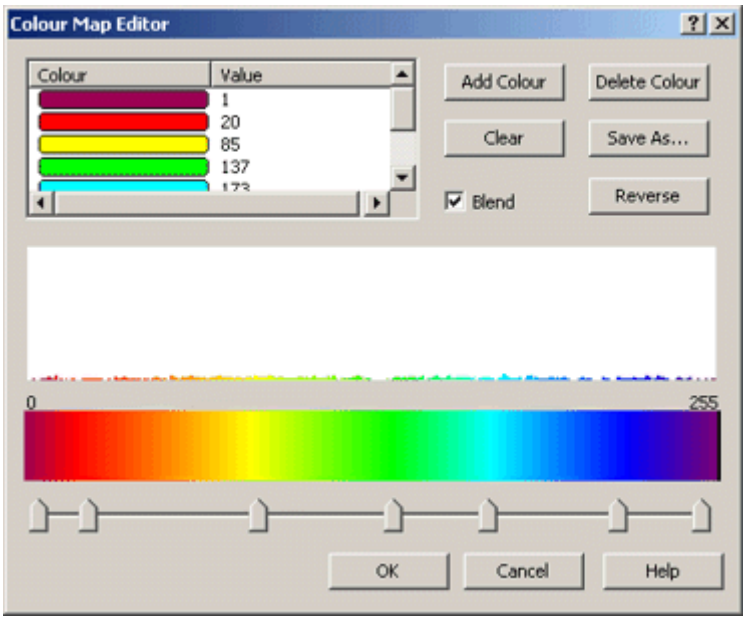
BASE Surface after filtering. Only depths between 0 metres and 18 metres are displayed

Example of filtering

11. Select the Range Filter check box to activate the filtering options.
12. Type the range of attribute values that displayed once the filter is applied.
13. Click **Apply** to implement changes.

Colour Map Editor

The Colour Map Editor dialog box allows you to modify colour maps and create new colour maps from existing maps.



The table on the top left section of the dialog box displays the colours and index values that represent the minimum depth for that the colour in the map.

The bottom part of the dialog box shows a histogram with the horizontal axis representing the colour range used in the image and vertical axis representing the number of pixels used to display the colour.

The first colour is the background and is always assigned to pixel value 1. The custom colour bar is filled from pixel values 1 to 255. Pixel value 0 is always assumed to be transparent.

Below the histogram is a slider control with each slider representing a colour in the map.

The dialog box also includes the following commands:

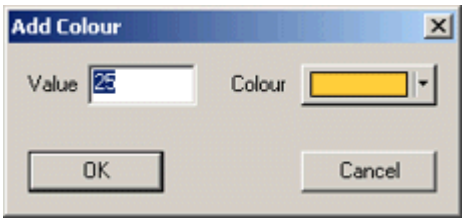
- **Add Colour**: Add a custom colour to the colour list.
- **Delete Colour**: Remove a newly selected colour.
- **Clear**: Remove a colour from the colour list.
- **Reverse**: Reverse the shallow and deep ends of the colour map.
- **Save As**: Save a new colour map.
- *Blend*: Merge colours together.

Change colour range

You can change the range of an individual colour using the table or slider control.

1. To change a colour range using the table, double-click a colour in the table.

The Add Colour dialog box is displayed.



2. Type a new number in the *Value* field and click **OK**.
3. Alternately, you can position the cursor on the slider representing the colour in the histogram and use the mouse to move the slider.

As the value is changed, the histogram and *Value* column in the table are refreshed to show the new colour value and range.

Add Colour

You can add or remove colours from an existing map, or create a new colour map.

1. Click **Add Colour** or click directly on the slide bar.

The Add Colour dialog box is displayed.

The number in the *Value* field shows the pixel value of the displayed colour.

2. Select a colour from the colour palette or click **Other** to create a custom colour from the standard Windows colour picker.
3. Click **OK**.

The selected colour is displayed in the dialog box.

4. Repeat the process to insert more colours.

As you select more colours, the colour values and the depths are displayed in a table. Each colour also has a slider control associated with it.

5. Move the slider to adjust the pixel value of that colour.

The *Value* and *Depth* data in the box adjusts as you move the sliders.

6. If you want to blend the colours, click the *Blend* check box.
7. If you want to remove a colour, click the colour value and click **Delete Colour**.
8. If you want to remove all the colours, click **Clear**.
9. Click **Save** to retain the settings as a colour map.

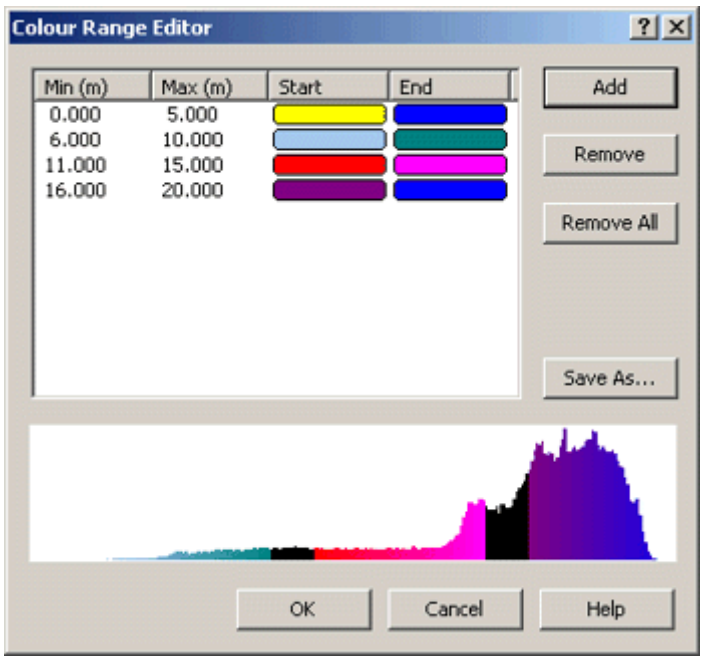
The Save As dialog box is displayed.

10. Type a file name for the colour map. All colour maps have a .cma extension and the directory always is ..\Hips\System.
11. Click **Save**.

The colour map is saved to the directory.

Colour Range Editor

The Colour Range Editor dialog box allows you to modify or create new colour range files.

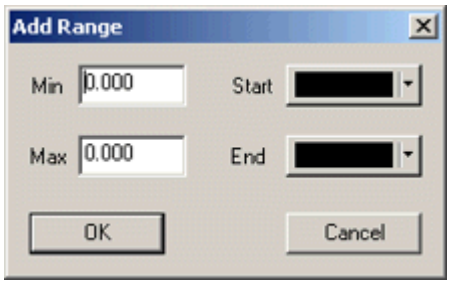


The top part of the dialog box contains a table showing the range of depths and the colours used to represent those depths.

The bottom part of the dialog box shows a histogram with the horizontal axis representing the colour range used in the image and vertical axis representing the number of pixels used to display the colour.

1. To add a colour range, click **Add**.

The Add Range dialog box is displayed.



2. Type the minimum and maximum values in the *Min* and *Max* fields.
3. Select a colour to represent the minimum and maximum values from the colour picker, or create a custom colour using the standard Windows colour palette.
4. Click **OK**.

The colour range is listed in the table and displayed in the histogram.

5. Repeat Steps 1 to 4 until all needed ranges are represented.
6. To remove a single colour range, select the range and click **Remove**.
7. To clear all colour ranges, click **Remove All**.
8. To save the ranges as a CRF for future use, click **Save As** to open the standard Windows Save As dialog box.
9. Click **OK** to close the Editor.

Update BASE Surfaces

Update BASE Surfaces created in HIPS 5.4 to HIPS 6.1 format.

BASE Surfaces created in HIPS 5.4 can only display attribute units in metres.

The BASE Surfaces are updated using a command-line utility (UpdateBASEunits.exe) located in the Bin directory. After the surfaces are updated, the unit settings in the Options dialog box can be applied.

1. From the Start button in the main Windows taskbar, select Start > Programs > Accessories > Command Prompt.

The Command Prompt is displayed.

2. Type the utility name (UpdateBASEunits) and the path where the surfaces are located. For example,

```
C:\Caris\HIPS\61\Bin\updatebaseunits C:\Caris\HIPS\61\Fieldsheets\Old
```

3. Press <Enter>

The BASE Surfaces are updated.

BASE Surface Commands

Open

Menu	File > Open Field Sheets

Open a saved Field Sheet.

1. Select the Open Field Sheets command.

The Open Field Sheets dialog box is displayed.

2. Select one or more field sheet files.
3. Click **Open**.

The field sheet file is displayed in the Session tab of the Control Window and the field sheet outline is shown in the Display window.

4. Click the Expand (+) icon to expand the field sheet file tree.
5. Select the BASE Surface check box.

The BASE Surface is now visible in the Display window.

You can switch the BASE Surface to on or off by clicking the check box.

Close

Menu	File > Close Field Sheets
Pop-up	Close

Close one or all open field sheets.

1. Select a field sheet file in the Control Window
2. Select Close from the right-click menu, or,
3. Select the Close Field Sheets command to close all open field sheets.

The field sheet(s) and associated BASE Surface(s) are closed. The field sheet outline is removed from the Display window and the file names are no longer displayed in the Control window.

Recompute

The recompute function rebuilds a BASE Surface and regenerates the Surface image.

1. Select a Surface name in the field sheet file tree so it is highlighted.
2. Select the Recompute BASE Surface command.

Menu	Process > BASE Surface > Recompute
Tool	

Only the track lines used previously in the Surface are processed during rebuilding.

The Recompute operation checks to see if the surface makes use of HIPS data that is no longer present.

If data is missing, a dialog box lists what is missing. You then have the option of terminating the Recompute process, or continuing without the missing data.

NOTE: If you choose to continue, the replacement surface will have the missing data lines removed from its list of contributing lines.

Add to BASE Surface

The Add to BASE Surface command applies new track lines to an existing Surface. You can add specific lines to the Surface or you can let the program select the lines to be added automatically.

1. Select the track line layer in the Control window.
2. Select the track line(s) you want to add to the BASE Surface so it is highlighted.
3. Select the BASE Surface layer in the field sheet file tree so it is highlighted.
4. Select the Add to BASE Surface command.

Menu	Process > Add to BASE Surface
Tool	

The BASE Surface Wizard dialog box (Step Two) is displayed.

5. Fill in the fields as needed.
6. Click **Next**.

The BASE Surface Wizard dialog box (Step 3) is displayed.

7. Fill in the fields as needed (see "THE THIRD DIALOG BOX IS DISPLAYED." ON PAGE 410).
8. Click **Finish**.

The BASE Surface is updated to include new track lines.

Remove From BASE Surface

Remove a survey line from a BASE Surface.

1. Select the survey line in the Project tab of the Control window.
2. Select a BASE Surface layer in the Session tab of the Control window.
3. Select the Remove from BASE Surface command.

Menu	Process > Remove From BASE Surface
Tool	

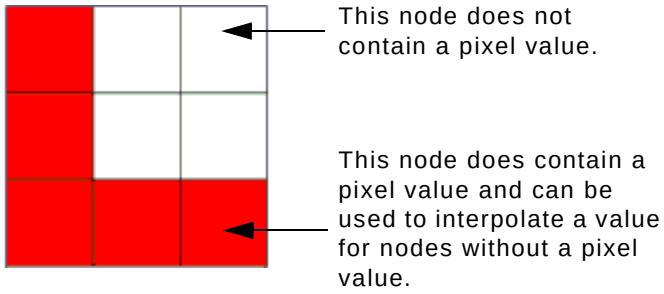
The BASE Surface is regenerated without data from the selected survey line.

Interpolate BASE Surface

The Interpolate Surface command eliminates small holes that appear in areas of sparse data. This sometimes happens in the outermost beams along the outside edges of surveyed areas where there is little or no overlapping coverage.

This command examines each BASE Surface node to determine if it contains a pixel value. If the node does not contain a value, the neighbouring pixels are examined to determine if enough of them contain pixel values to justify interpolating a value for the hole. This limits the interpolation to holes in the BASE Surface and prevents it from expanding the surface outward from the survey area. A copy of the BASE Surface, with these gaps filled, is then created.

In the following example, in a 3 x 3 BASE Surface node area, five nodes have pixel values (shown in red).

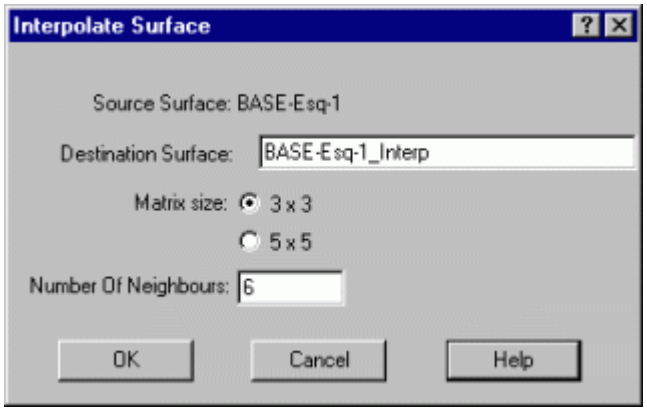

This node does not contain a pixel value.
This node does contain a pixel value and can be used to interpolate a value for nodes without a pixel value.

This means if you selected the *Number of Neighbours* value (see dialog box below) as being five or less, a pixel value is created for the node without a value from the neighbouring nodes.

1. Select a surface layer in the Control window so it is highlighted.
2. Select the Interpolate Surface command.

Menu	Process > BASE Surface > Interpolate
Pop-up	Interpolate BASE Surface

The Interpolate Surface dialog box is displayed.



The *Destination BASE Surface* field displays the file name of the copy that is created from the original surface. The default name is the original BASE Surface name with _Interp appended to it. You can change the name, if necessary.

3. [Optional] Type a new name in the *Destination BASE Surface* field.

The *Matrix Size* determines the number of nodes closest to the node that can be used to interpolate a pixel value.

4. Select either the *3 x 3* or the *5 x 5* option.

The *Number of Neighbours* field is a threshold level used to determine the minimum pixels in the matrix area that must be present to interpolate a new pixel value. The matrix size determines the maximum and minimum values for the neighbours field. For example, if you select 3 x 3 for the matrix size, then the possible maximum number of neighbouring nodes with pixels is 9 and the minimum number is 3.

5. Type the number of neighbouring nodes used to interpolate pixels.
6. Click **OK**.

A copy of original BASE Surface is created and displayed in the field sheet data tree in the Control window. Note that the interpolated BASE Surface cannot be updated with the Automatic Update command in the Options dialog box.

As you can continue to re-interpolate the Surface, the effects accumulate to remove any remaining gaps. To do this, select the interpolated surface in the Control window and select the Interpolate Surface command again. The second interpolation is applied to the selected surface layer, you cannot create a new BASE Surface layer.

Every time an interpolated BASE Surface is re-interpolated, the parameters are stored. Therefore, when using the Rebuild command (see "RECOMPUTE" ON PAGE 423) on an interpolated BASE Surface, all interpolations are performed on the BASE Surface layer.

Delete

Permanently delete a BASE Surface from a field sheet.

1. Select a BASE Surface layer in the Control Window.
2. Right-click and select the Delete command from the pop-up menu.

Pop-up	Delete

The BASE Surface layer is permanently deleted from the field sheet.

Finalize BASE Surface

The finalized BASE Surface is a finished version of the Surface that is ready for export or for further processing (for example, a Product Surface). The automatic update function in the Options dialog box (see "GENERAL" ON PAGE 75) cannot be applied to a finalized Surface if soundings are re-cleaned in the Swath or Subset Editors.

Any of the following three operations can be applied to finalize a BASE Surface:

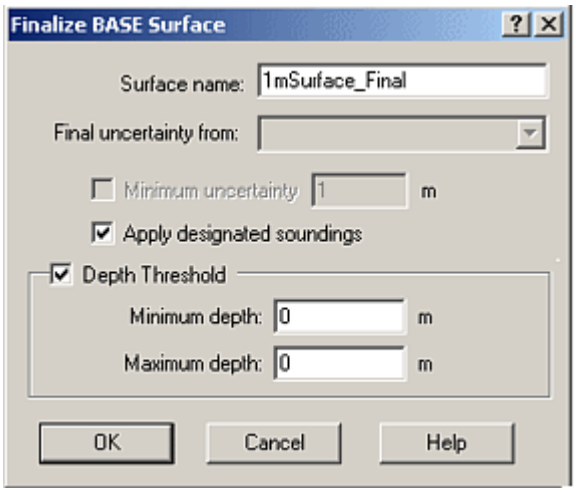
- **Minimum uncertainty**: Vertical uncertainty can fall to almost zero if a node contains too many soundings. This function applies a user-defined vertical uncertainty to each node to compensate for this situation.
- **Designated soundings**: If the dataset contains designated soundings, the Surface is regenerated and the designated soundings' depth values are applied to the nearest nodes. (See "DESIGNATED SOUNDINGS" ON PAGE 223).
- **Depth thresholds**: A Finalized BASE Surface can be modified to represent a range of depths levels in the field sheet. When the Finalized Surface is generated, only these depths are displayed.

Too create a finalized BASE Surface:

1. Select a BASE Surface layer in the Session tab of the Control window.
2. Select the Finalize BASE Surface command.

Menu	Process > BASE Surface > Finalize

The Finalize BASE Surface dialog box is displayed.



3. The default name for the Finalized Surface is the name of the source BASE Surface with _Final appended to the end. You can change this by typing a new name.

The values for calculating the uncertainty of the finalized surface can be selected from three sources:

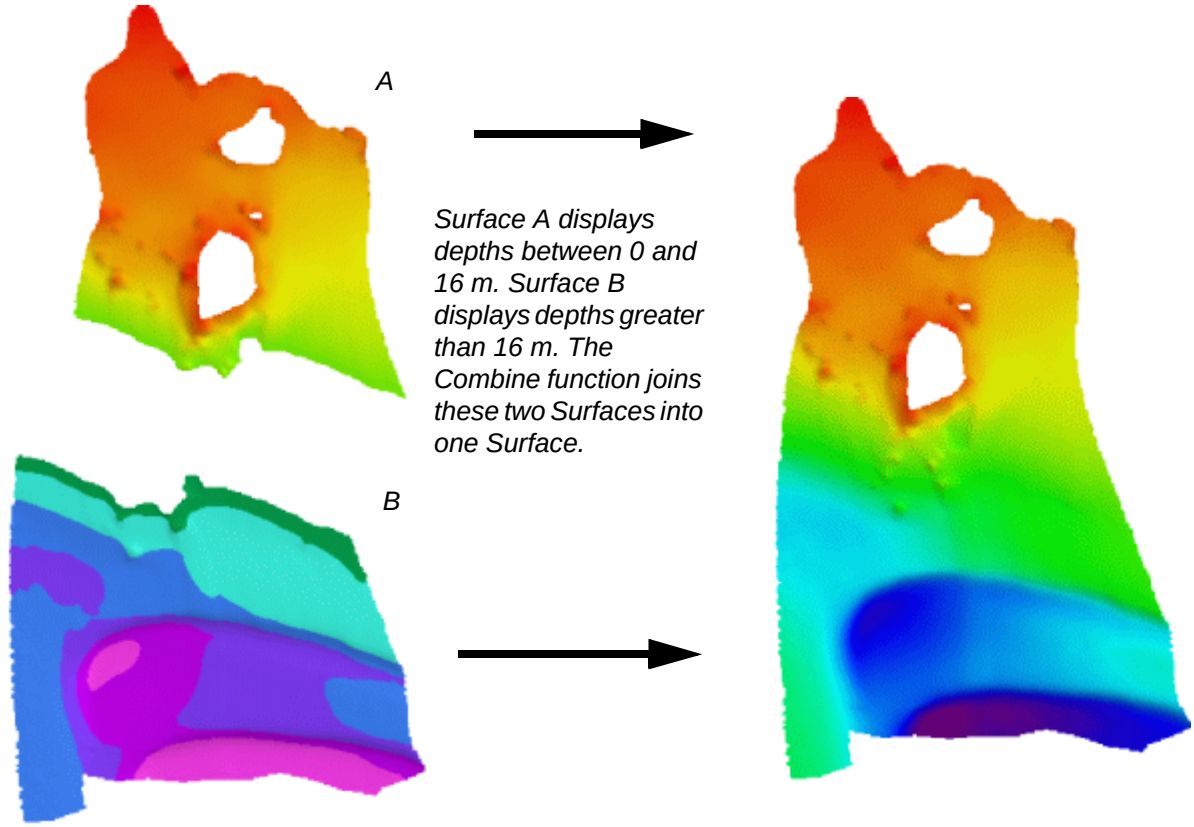
- *Uncertainty*: The vertical uncertainty values in the source Surface.
- *Std Deviation*: The standard deviation values from the source Surface (scaled to 95% confidence interval).
- *Greater of two*: The larger of the two above values.

4. Select the *Final Uncertainty* from the drop-down list (this option is dimmed if source Surface does not contain uncertainty values).
5. Select the *Minimum Uncertainty* check box to enable this option.
6. Type a depth value to represent the minimum uncertainty.
7. Select the *Apply Designated Soundings* check box to use this option.
8. Select the *Depth Threshold* check box to create a finalized Surface that displays only node depths between the minimum and maximum depth values.
9. Type the minimum and maximum depth values in their respective fields.
10. Click **OK**.

A finalized BASE Surface is generated and visible in the Display window. The finalized BASE Surface is listed in the Session tab of the Control window.

Combine

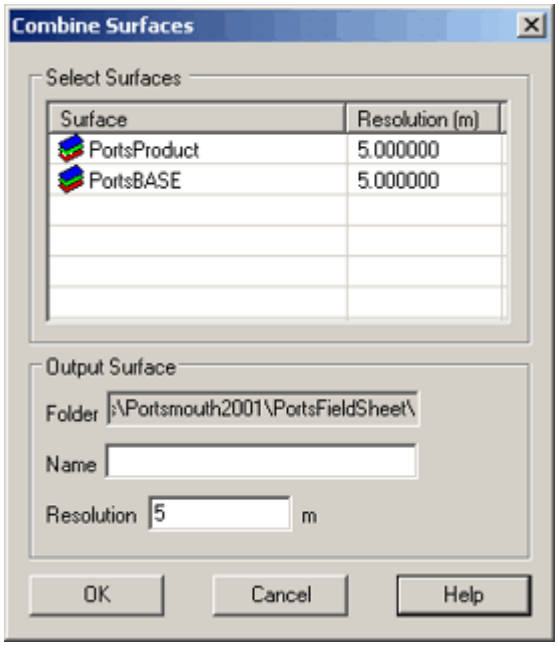
The Combine options stitches two or more BASE Surfaces together to form one surface. This option is used when you have created finalized BASE Surfaces with different depth thresholds (see "FINALIZE BASE SURFACE" ON PAGE 427), or when you have two adjacent Surfaces with different resolutions. The following diagram demonstrates how the Combine function works.


A
B
Surface A displays depths between 0 and 16 m. Surface B displays depths greater than 16 m. The Combine function joins these two Surfaces into one Surface.

1. Select field sheet in the Session tab of the Control window that contain the BASE Surfaces you want to combine.
2. Select the Combine command.

Menu	Process > BASE Surface > Combine
Pop-up	Combine (Control window only)

The Combine Surfaces dialog box is displayed.



3. Select the BASE Surfaces from the file tree by two methods (you may have to click the expand icon to view all the available Surfaces):
   - Press <Ctrl> and select individual surfaces with the mouse.
   - Press <Shift> and select a range of surfaces by clicking the first and last surface in the range. All surfaces between the first and last surface are selected.
4. The Output Folder field is read-only and displays the directory path of the selected field sheet. The combined BASE Surface is saved to this folder.
5. Type a name for the combined Surface.
6. Type a resolution for the combined Surface
7. Click **OK**.

A new BASE Surface that combines the selected Surfaces is generated. The combined Surface is listed in the Session tab of the Control window. The combined BASE Surface reproduces all the attribute layers contained in the source Surfaces.

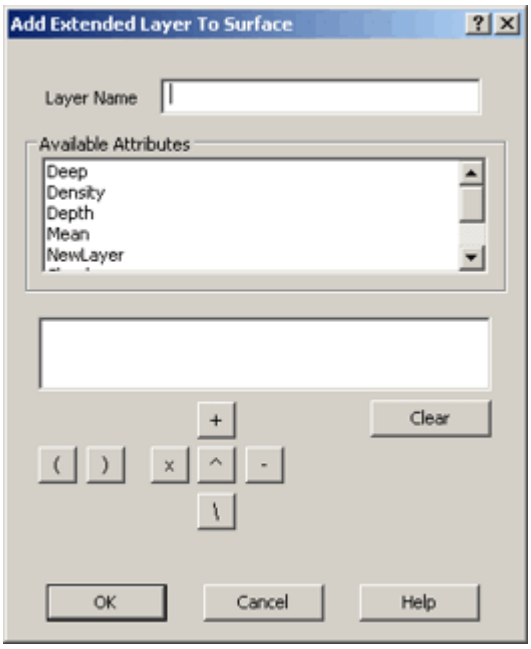
New Layer

Create a new sub-layer in an existing BASE Surface. The layer is created using existing attributes and mathematical operators to create SQL-style queries.

1. Select a BASE Surface layer in the Control window.
2. Select the command.

Pop-up	New Layer (Control window only)

The Add Extended Layer to Surface dialog box is displayed.

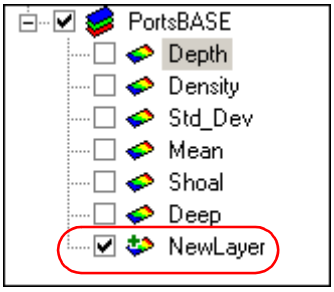


3. Type a name for the new surface layer in the *Layer Name* field.

The following steps create the criteria for the new surface. The criteria is determined by attributes and mathematical operators. You can insert as many attributes and operators needed to create the criteria.

4. Double-click any of the attributes in the *Available Attributes* list to insert the attribute into the field below.
5. Click any of the operators to add it to the field.
6. To remove the criteria from the field, click **Clear**.
7. Click **OK**.

The new layer is drawn in the Display window and is listed as a BASE Surface sub-layer in the Control window.



QC Report

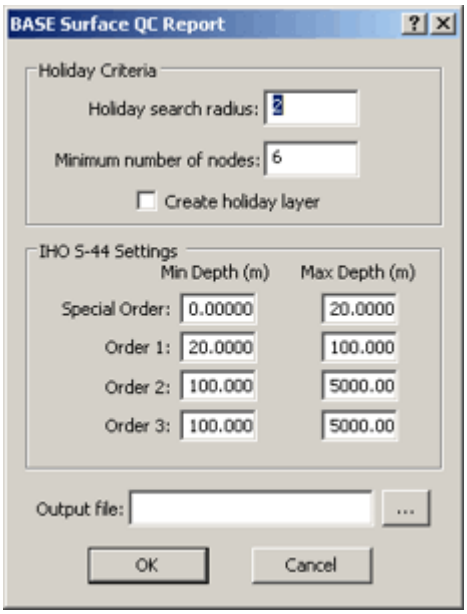
The BASE Surface Quality Control report lists significant data gaps (“holidays”) within the surface and analyzes how much of your data conforms to the S-44 standard for survey orders.

1. Select a BASE Surface layer in the Control window.
2. Select the QC Report command.

QC Report

Menu	Process > BASE Surface > QC Report
Pop-up	QC Report (Control window only)

The BASE Surface QC Report dialog box is displayed.



The top section of the dialog box sets the criteria for finding significant data gaps (known as “holidays”). A holiday is formed when enough gaps are clustered together within a specified radius. A layer that displays these holidays can also be created. This layer does not display every gap in the surface, but only those gaps that meet the criteria.

1. Type the number of nodes to use as a radius for searching for gaps in the *Holiday Search Radius* field.

2. Type the *Minimum number of nodes* that must contain data so that a holiday is not formed. For example, if the value is 6, then a node without data must be surrounded by five or less nodes with data to be included (with other empty nodes) in a holiday.
3. Select the *Create holiday layer* check box to create a layer displaying the data holidays selected by the search criteria.

The bottom section compares uncertainty values contained in the surface to the S-44 standard. This determines how many nodes within the surface meet the S-44 criteria.

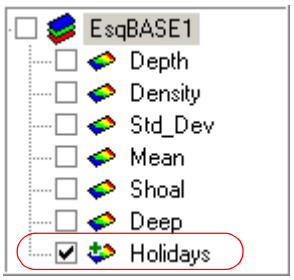
The depth levels shown in the dialog box are the minimum and maximum depth levels for each survey order. You can change these to conform with depths in your surface.

4. [Optional] Type new depth levels for any of the survey orders. You can have overlapping depth levels.

A text file of the QC report can be saved to a text format and viewed later.

5. Click **Browse** to select a directory path and name for the QC report. The path and name is shown in the *Output File* field.
6. Click **OK**.

The report is shown in the Output window and, if selected, saved to a text file. The Holiday layer is displayed in the Control window.



Holiday sub-layer for a BASE Surface

The following section describes the sections in the QC report file.

QC Report File The following is an example of a QC report file.

```
BASE Surface QC Report
----------------------
Date and Time: 11/9/2006 10:59:09 AM
Surface: d:\hips\FieldSheets\HIPSTraining\smallsheet\asdfasdf.hcs
Holiday Search Radius: 1
Holiday Minimum Number of Nodes: 6
Holiday layer created: Yes

Number of nodes processed: 6,181
Number of nodes populated: 6,131
Percentage of nodes populated: 99.191
Number of holidays detected: 5
IHO S-44 Special Order:
     Range: 0.0 to 10.0
     Number of nodes considered: 620
     Number of nodes within: 0
     Residual mean: 0.307
S-44 Order 1:
     Range: 10.0 to 100.0
     Number of nodes considered: 5,511
     Number of nodes within: 0
     Residual mean: 0.035
S-44 Order 2:
     Range: 10.0 to 5000.0
     Number of nodes considered: 5,511
     Number of nodes within: 5,511
     Residual mean: -0.484
No depths within IHO S-44 Order 3
```

The first section contains information about the report file and the options the data holiday options set in the dialog box:

- The data and time that the QC report file was created.
- The BASE Surface on which the QC report is based
- The holiday search radius.
- The minimum number of nodes required to omit a node (without data) from being included in a holiday.
- If a holiday layer was created or not.

The second section contains information on the number of nodes containing data:

- The total number of nodes processed in the surface
- The total number of nodes that contained data
- Percentage of the surface that contains data.

- Number of data holidays located.

The third section contains information about each S-44 survey order:

- The depth range for the survey order (set in the dialog box).
- The number of nodes within that depth range with uncertainty values that were considered for the survey order.
- The number of nodes that fall within the depth error limits for that survey order.
- The residual mean is derived from the computed and reported uncertainty. If the value is negative, the uncertainty values meet the requirements for the selected survey order. If the value is positive, the uncertainty values do not meet the requirements for the selected survey order. Therefore, if no nodes were considered for the survey order, the residual mean would have to be above zero.

Surface Filtering

Surface filtering uses a scaled standard deviation or uncertainty threshold to reject soundings that fall outside that threshold.

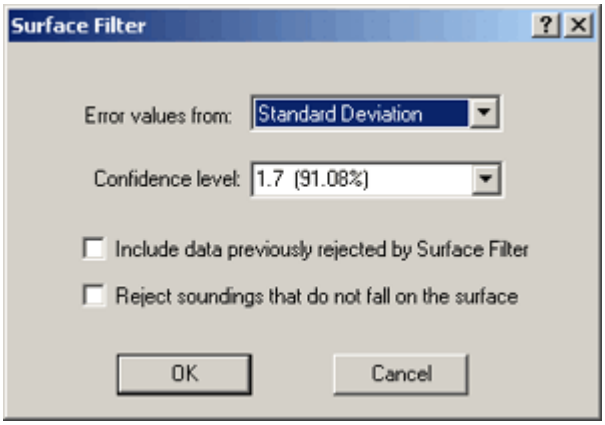
Soundings that fall outside the threshold are assigned a Rejected by Surface Filter flag and are no longer available for processing (unless they are given an Accepted flag again).

1. Select a BASE surface layer in the Control window.
2. Select the Surface Filter command.

r

Menu	Tools > Apply Filters > Surface Filter

The Surface Filter dialog box is displayed.



3. Select the error values from either *Standard Deviation* or *Uncertainty* to set the threshold for the filter.
4. Select the *Confidence Level* from the drop-down list to scale the standard deviation or uncertainty thresholds.
5. Select the check box to include rejected data when running the filter.
6. Select the check box to reject soundings that are offset from the surface.
7. Click **OK**.

18

CUBE Processing

Introduction

The Combined Uncertainty and Bathymetry Estimator (CUBE) generates a surface that contains multiple hypotheses representing potential depth variances on the seafloor.

After a CUBE Surface is generated, you can examine the hypotheses in the Subset Editor to determine if they are valid or not. The hypotheses can be replaced by alternative hypotheses (if necessary). A filtering tool is provided to reject soundings that fall outside specified parameters and another CUBE Surface can then be generated. Thus, CUBE is used in an iterative process to filter data to produce the best-possible representation of the surveyed area.

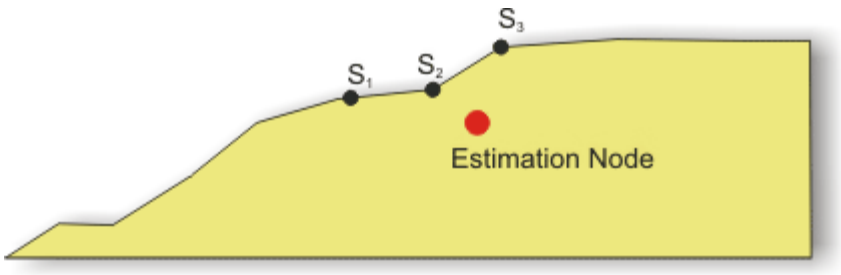
Horizontal and vertical uncertainty values are necessary to create BASE Surfaces using CUBE.

The CUBE process uses sounding propagation along with disambiguation to create and select hypotheses.

When soundings are propagated to a grid of estimation nodes:

- Soundings with a low vertical uncertainty are given more influence than soundings with high vertical uncertainty
- Soundings with a low horizontal uncertainty are given more influence than soundings with a high horizontal uncertainty.
- Soundings close to the node are given a greater weight than soundings further away from the node.

Generally, as soundings are propagated to a node, a hypothesis (depth value) is developed at that node. If a sounding's value is not significantly different from the previous sounding then the same or modified hypothesis is used. If the value does change significantly, a new hypothesis is created. A node can contain more than one hypothesis.



In the above graphic, two soundings—S_1 and S_2—have similar values and therefore are part of the same hypothesis at the

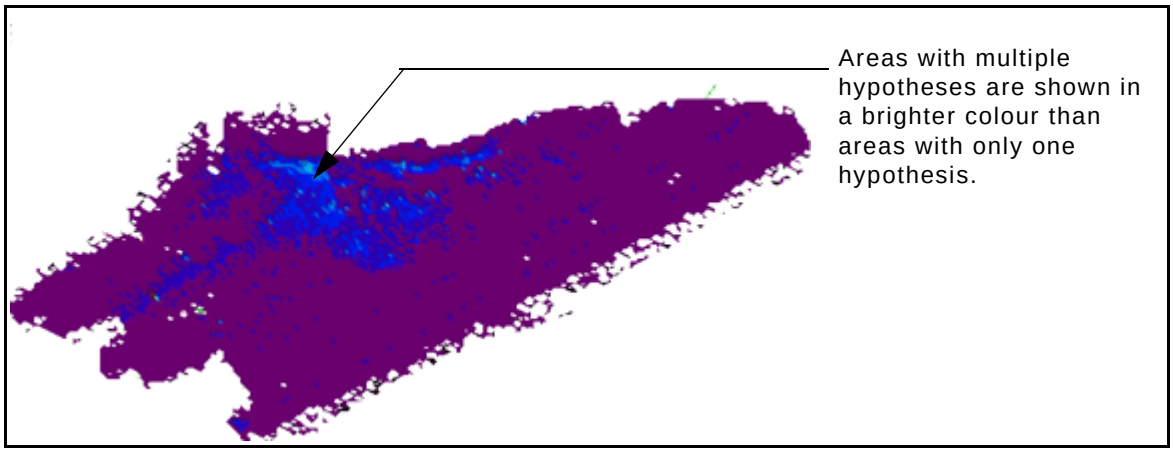
estimation node. However, sounding three (S_3) has a significantly different value so it forms a new hypothesis.

The final process in CUBE is disambiguation. Disambiguation selects one hypothesis over others. There are four disambiguation options:

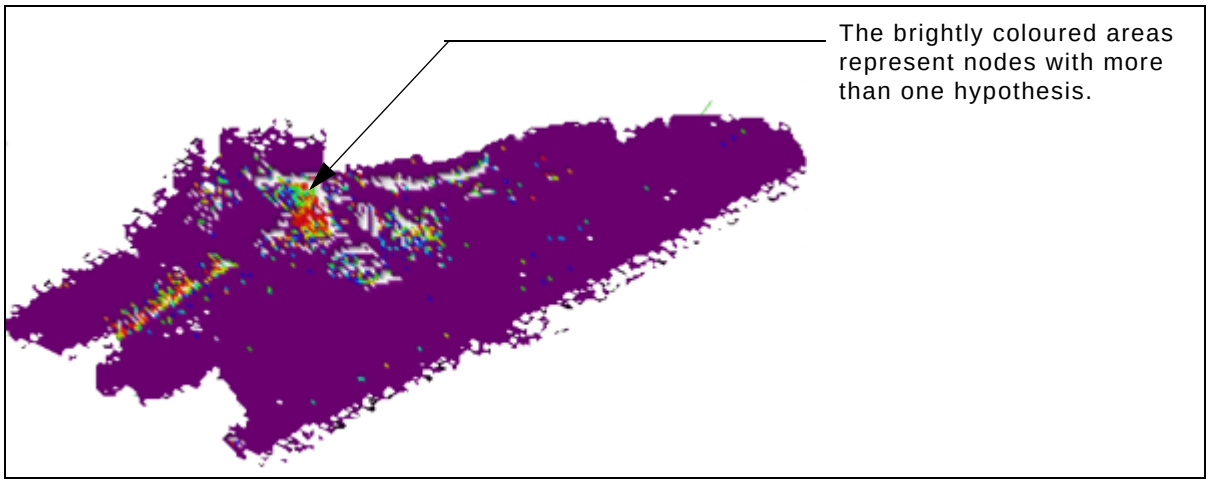
- Density: Select the hypothesis with the greatest number of sounding samples.
- Locale: Select the hypothesis that is most consistent with the surrounding nodes that have only one hypothesis.
- Locale and Density: Select the hypothesis the contains the greatest number of soundings and is also consistent with neighbouring nodes.
- Initialization: Select the hypothesis that is nearest to a node value of a previously created BASE Surface. Initialization differs from the other methods because it filters potential outlier soundings just prior to disambiguation.

When a surface using the density and locale options is generated, two layers specific to the CUBE Surface are displayed.

- The Hypothesis Count layer is a visual representation of hypothesis density at a node. A Surface with a Hypotheses Count layer is displayed below.



- The Hypothesis Strength layer is a visual representation of the mathematical confidence of a chosen hypothesis. Each node is given a value ranging from 0.0 (high confidence) to 5.0 (low confidence). Nodes with one hypothesis have a confidence value of 0.0 while nodes with multiple hypotheses will have confidence values greater than 0.0. A Hypothesis Strength layer is displayed below.



More than one hypothesis or a low confidence value does not necessarily mean an error. Uneven areas (slopes, for example) show more than one hypothesis because of the changing terrain. Nodes with multiple hypothesis should be examined in the Subset Editor. See "HYPOTHESES CLEANING" ON PAGE 454.

When you select the Initialization method of disambiguation, two additional layers are created:

- Guide_Depth: Depths from the initialization surface.
- Guide_Uncertainty: Vertical uncertainty values from the initialization surface.

The User Nominated layer displays the nominated hypotheses that were chosen over the hypotheses selected by CUBE disambiguation (see "NOMINATE ALTERNATIVE HYPOTHESES" ON PAGE 452).

Generating CUBE Surfaces

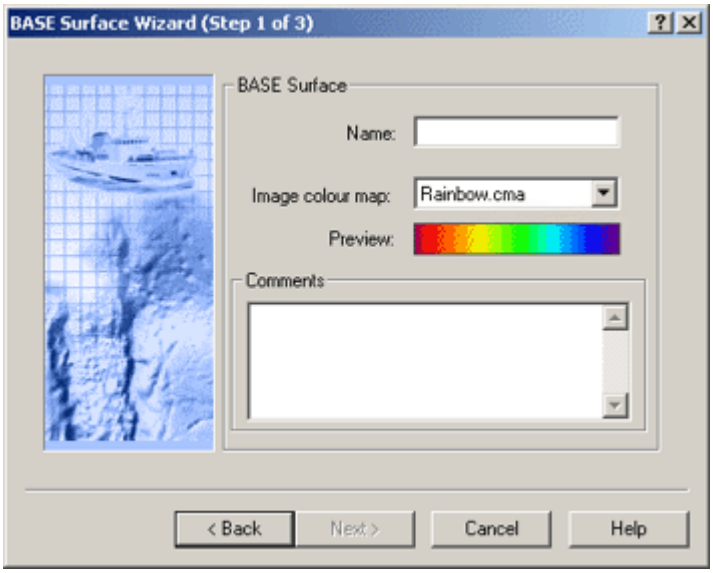
CUBE Surfaces are created using the BASE Surface wizard. All track lines must be merged and TPE must be computed before a CUBE Surface can be created.

1. Select the field sheet area for the CUBE Surface:
   - To create a CUBE Surface for the entire field sheet area, select a field sheet in the session data tree in the Control window.
   - To create a CUBE Surface for a single track line, select a track line in a field sheet and then select the field sheet layer in the data tree.
2. Select the New Base Surface command.

Menu	Process > BASE Surface > New
Tool	

CUBE Surface (Step 1)

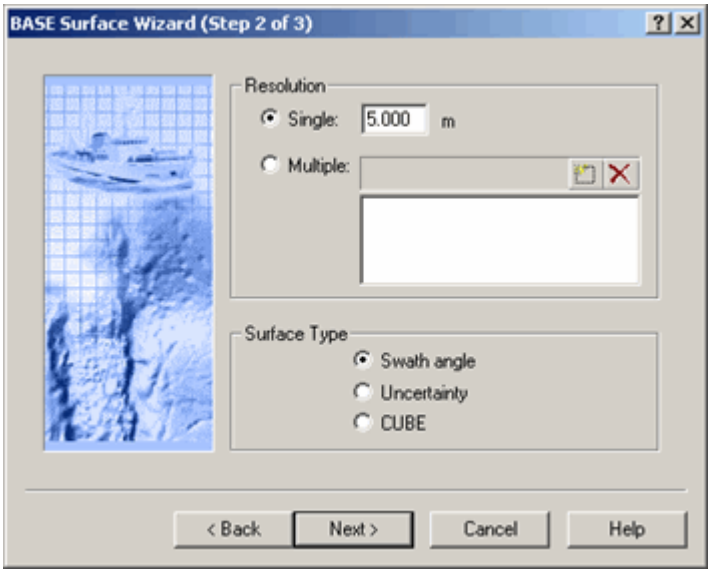
The BASE Surface Wizard (Step 1) dialog box opens.



1. Type a *Name* for the BASE Surface.
2. Select an *Image Colour Map* file for the TIFF image that is created after the Surface has been computed. You can change the colour map later.
3. Type any *Comments* (if necessary) in the text box.
4. Click **Next**.

CUBE Surface (Step 2)

The BASE Surface Wizard (Step 2) dialog box is displayed.



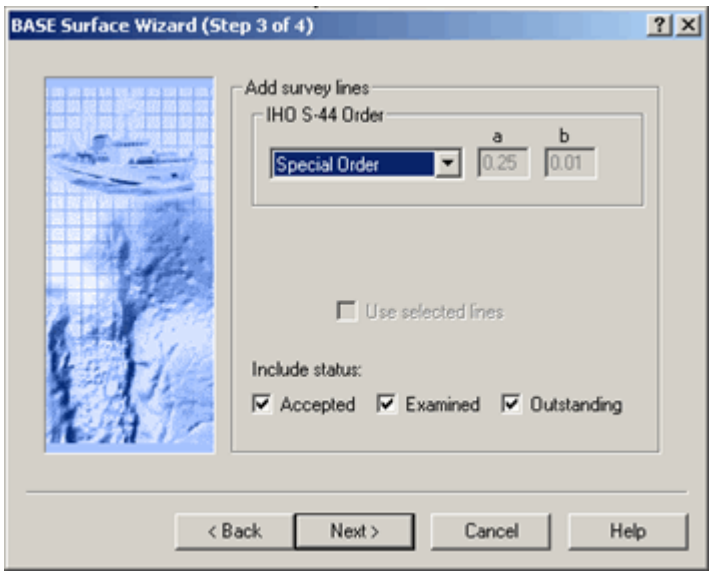
The *Resolution* value(s) sets the distance(s) between CUBE Surface nodes. You can use a single resolution for the entire surface, or, use multiple resolutions for a range of depths.

To enter a range of depth values, you must select the *Multiple* option. Values are entered in the following format: depth-depth,resolution. For example 0.0-10.0,5

5. Select the *Single* field and enter a resolution value, or select the Multiple option and enter a range of values.
6. Select *CUBE*.
7. Click **Next**.

CUBE Surface (Step 3)

The BASE Surface Wizard (Step 3) dialog box is displayed.



A sounding's area of influence (the number of nodes to which a sounding is applied) is determined by the vertical uncertainty limit in a selected S-44 survey order. When a sounding's vertical uncertainty increases beyond the requirement for the survey order then it cannot contribute to the node.

1. Select an *IHO S-44 Order* from the drop-down list.

The *a* (constant depth error) and *b* (factor of depth dependent errors) fields are automatically filled when a survey order is selected. The values are read from ..\HIPS\System\IHO_Standards.xml.

If one or more track lines were selected before the BASE Surface process was started, the *Use Selected Lines* check box is enabled.

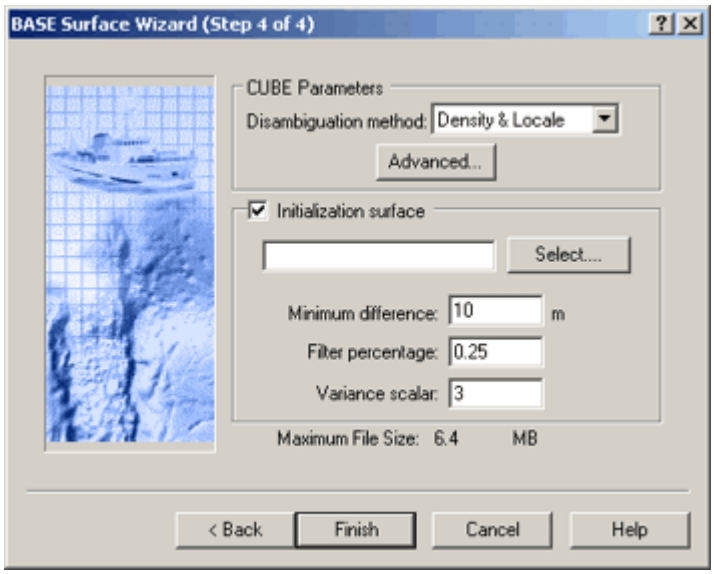
2. Clear this check box if you want to apply the BASE Surface to the field sheet area.

The *Accepted* check box is selected by default.

3. Select either the *Examined* or *Outstanding* check boxes to include data with these status flags in the BASE Surface process.
4. Click **Next**.

CUBE Surface (Step 4)

The BASE Surface Wizard (Step 4) dialog box is displayed.



Disambiguation is the process of selecting hypotheses at the nodes. There are four disambiguation options:

- Density: Select the hypothesis with the greatest number of sounding samples.
- Locale: Select the hypothesis that is most consistent with its surrounding nodes that has only one hypothesis.
- Density & Locale: Select the hypothesis that contains greatest number of soundings and is also consistent with neighbouring nodes.
- Initialization: Select the hypothesis that is closest to an initialization surface (i.e., a previously created BASE Surface).

5. Select a *Disambiguation* option from the drop-down list.
6. Click **Advanced** to view more options for CUBE display. For more information, see "ADVANCED OPTIONS" ON PAGE 446.

The *Initialization Surface* check box activates filtering tests to remove potential outliers from CUBE Surface processing. If you select the *Initialization* disambiguation method then you must complete the following fields. Otherwise, select *Initialization Surface* as an optional filtering method used in conjunction with other disambiguation methods.

7. Select the *Initialization Surface* check box to activate the filtering options.

8. Type the path and name of an existing BASE Surface, or click **Select** and choose the BASE Surface.

Initialization Surface filtering uses depth and vertical uncertainty values in an existing BASE Surface (the Initialization Surface) to exclude potential outliers from the new surface. There are three filtering tests:

- *Minimum Difference* is a set distance from an existing BASE Surface node. For example, if the *Minimum Distance* is set at 10 metres, all soundings that are not within 10 metres of the node are excluded when a new Surface is generated.
- *Filter Percentage* is a percentage of depth at a node. For example, if the depth at a node is 20 metres and the percentage value is 0.25, all soundings that are more than 5 metres from the node (20x0.25) are excluded from the new Surface.
- *Variance Scaler* is a multiplier of vertical uncertainty at a node. For example, if the scaler is set to three and depth uncertainty for a node is one metre then soundings more than three metres (1x3) from the node are excluded from the processing of the new BASE Surface.

The shoalest values from the three tests are then used as the threshold to filter outlier soundings.

9. Select any of the three tests to filter soundings during disambiguation.
10. Click **Finish**.

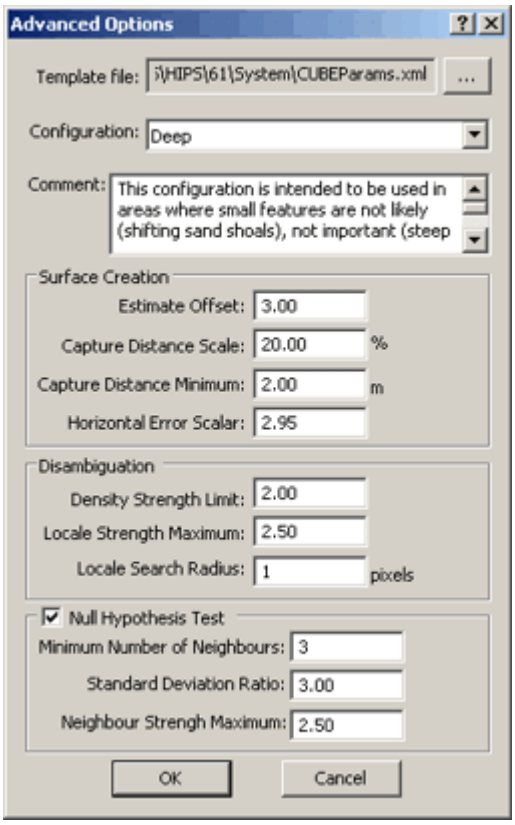
A CUBE Surface is shown in the Display window and the layers associated with the Surface are listed in the Control window. See "VIEW BASE SURFACES" ON PAGE 412 for more information on display options.

Advanced Options

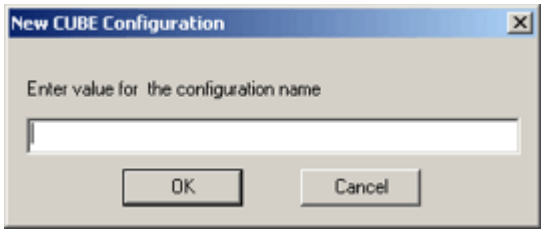
The Advanced Options dialog box displays more detailed configuration settings for the CUBE surface. These settings are contained in the CUBEparams.xml file. You can use the settings in this file or select a custom configuration file. If you use a custom file, make sure that you use the same syntax and structure as the CUBEparams.xml file.

Changes made in this dialog box are saved to the CUBEparams.xml file or to the configuration file you selected.

When the **Advanced** button is clicked in the CUBE Surface (Step 4) dialog box, the Advanced Options dialog is displayed.



1. [Optional] Click **Browse** to select another configuration file. The file path is displayed in the *Template File* field.
2. Select a configuration setting from the *Configuration* drop-down list. If you select New, the New CUBE Configuration dialog box is displayed.



3. [Optional] Type the name for the new configuration setting and click **OK**.
4. [Optional] Type any additional information in the *Comments* field.
5. Complete the following fields, as needed:
   - *Estimate Offset*: The threshold for significant offset from current estimate to warrant an the creation of a new hypothesis. The value must be between 0.1 and 10.0.
   - *Capture Distance Scale*: Scale on predicted or estimated depth for how far out to accept data. Value is a percentage of depth used to limit the radius of influence a sounding may have on the grid. Value must be between 1.00 and 100.00.
   - *Capture Distance Minimum*: The minimum value (in metres) on predicted or estimated depth for how far out to accept data. This value is used in conjunction with Capture Distance Scale to limit the radius or influence of a sounding. Value must be between 0.0 and 100.00.
   - *Horizontal Error Scalar*: The value used to scale the horizontal error of each sounding when used in the radius of influence computation. Value must be between 0.0 and 10.00.
   - *Density Strength Limit*: The strength value used to switch from the 'density' disambiguation method to the 'locale' version when using the density & locale algorithm. Value must be between 0.00 and 5.00.
   - *Locale Strength Maximum*: The maximum strength value allowed as part of the mean in the locale algorithm. Value must be between 0.00 and 5.00.
   - *Locale Search Radius*: The radius of the search when computing the trimmed mean. The values are in pixels and must be greater than zero. The original system default value is 1.
   - *Null Hypothesis Test*: Flag used to control the application of the NULL hypothesis test. Value must be either True or False. Default value is False.
   - *Minimum Number of Neighbours*: During the Null Hypothesis test, this controls the minimum number of neighbours that a node must have in order to be considered for the Standard Deviating Ratio test. If the node has less the specified number, it is automatically marked as 'Null'. Values must be between 2 and 8.
   - *Standard Deviation Ratio*: During the Null Hypothesis test, this represents the cut-off limit for the standard devotion ratio including the node in question to the standard deviation of the qualified

neighbouring nodes. If the computed ratio exceeds the specified value, the node is marked as 'Null'. Values must be between 0.0 and 10.0.

- *Neighbour Strength Maximum*: The maximum strength value that is allowed to be considered as part of the standard deviation computations in the NULL hypothesis test. Value must be between 0.00 and 5.00.

6. Click **OK** to save and apply settings.

Hypothesis Editing

CUBE surfaces can be displayed in the Subset Editor for further examination and editing.

r

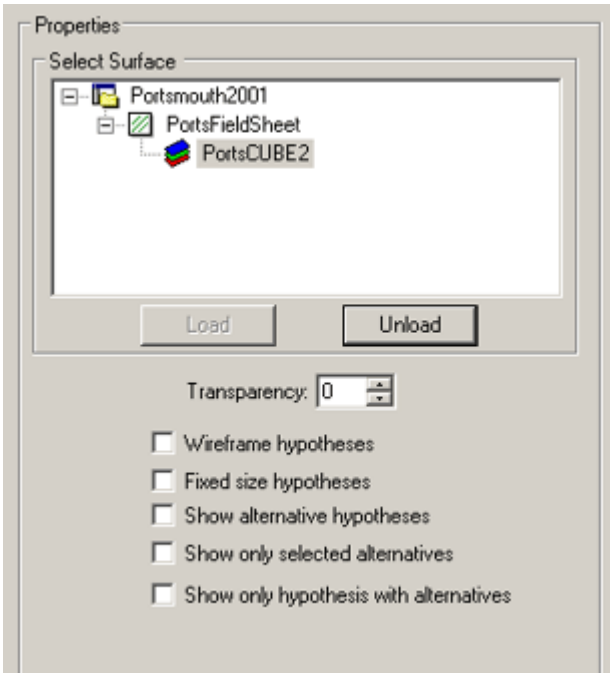
Menu	Tools > Subset Editor
Tool	
Pop-up	Tools > Subset Editor

1. Select the r command.
2. Use the mouse to adjust the size of the Subset Editor bounding box by moving the handles.

Data contained in the bounding box area is loaded into the 3-D Subset view.

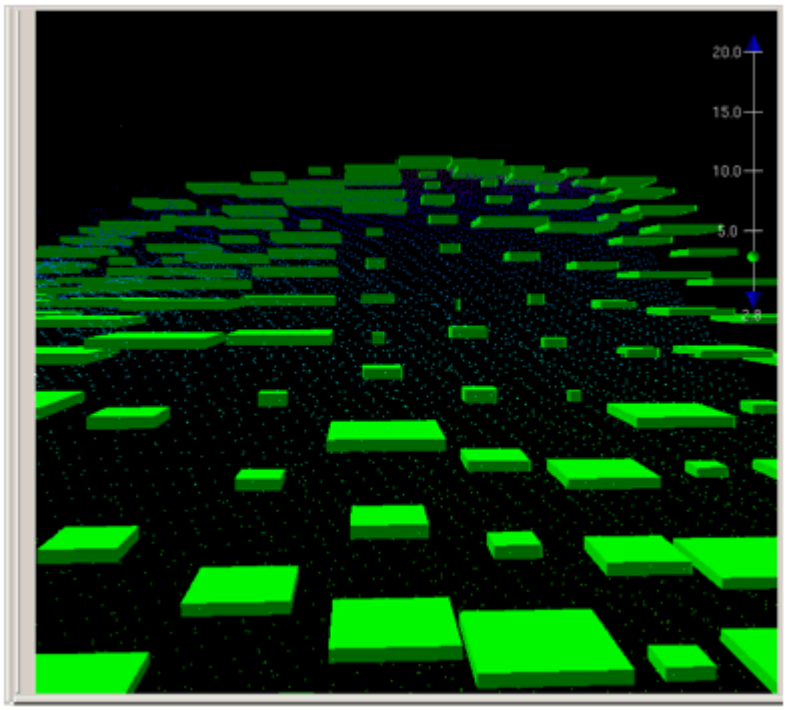
3. In the Subset Editor tab, select the CUBE Hypothesis layer.

The property options for the CUBE surface are displayed.



4. Expand the data tree and select a BASE Surface containing CUBE data.
5. Click **Load** to view the CUBE surface.
6. To return to the normal display in the 3-D and 2-D Views, click **Unload**.

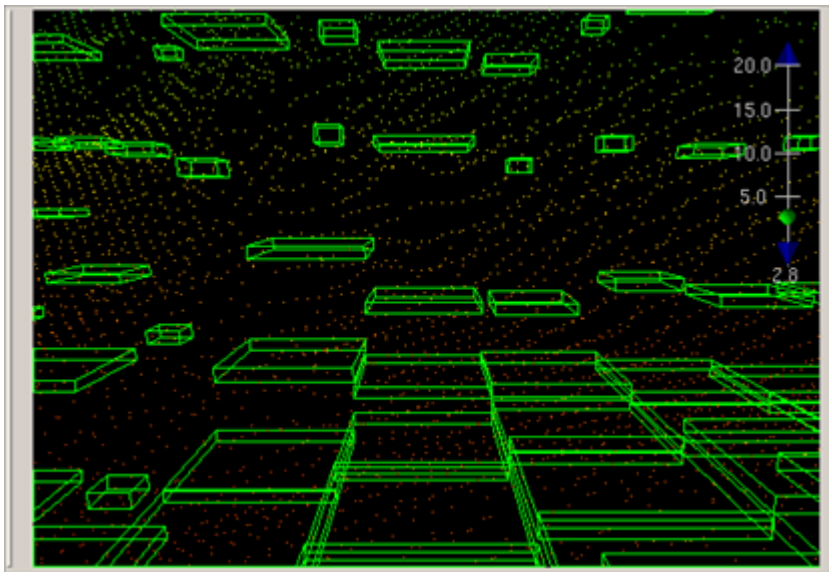
Below is an example of a section of a CUBE surface in the 3-D View.



Hypotheses are represented as squares. Squares are shown with different dimensions to reflect the confidence level given to each hypothesis. These confidence levels range from 0.0 (the highest value) to 5.0 (the lowest value). The size of a square is related to the numerical confidence level: large squares represent nodes with the high confidence values while smaller squares represent nodes with low confidence values.

The thickness of the square represents the vertical uncertainty assigned a node (to a 95% confidence interval). The thicker the square, the greater the vertical uncertainty.

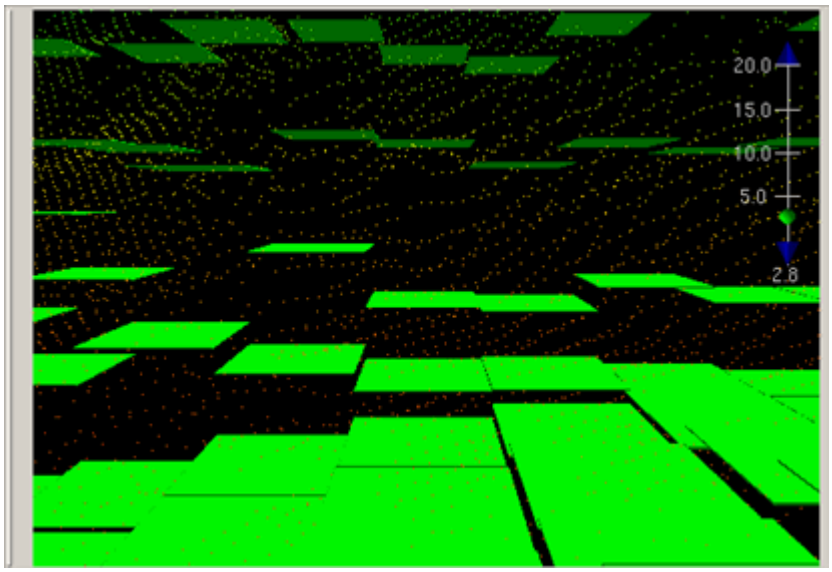
Hypotheses squares can also be displayed without a fill so that they are transparent except for the outlines of the squares.



7. Select the *Wireframe Hypotheses* check box.

The 3-D View is refreshed to display the squares without fill.

The *Fixed Size Hypotheses* option flattens the hypotheses squares and redraws them to a standard size. This option is useful if you want to examine the dataset for holes or to see the exact position of a square in the subset.



8. Select the *Fixed Size Hypotheses* check box.

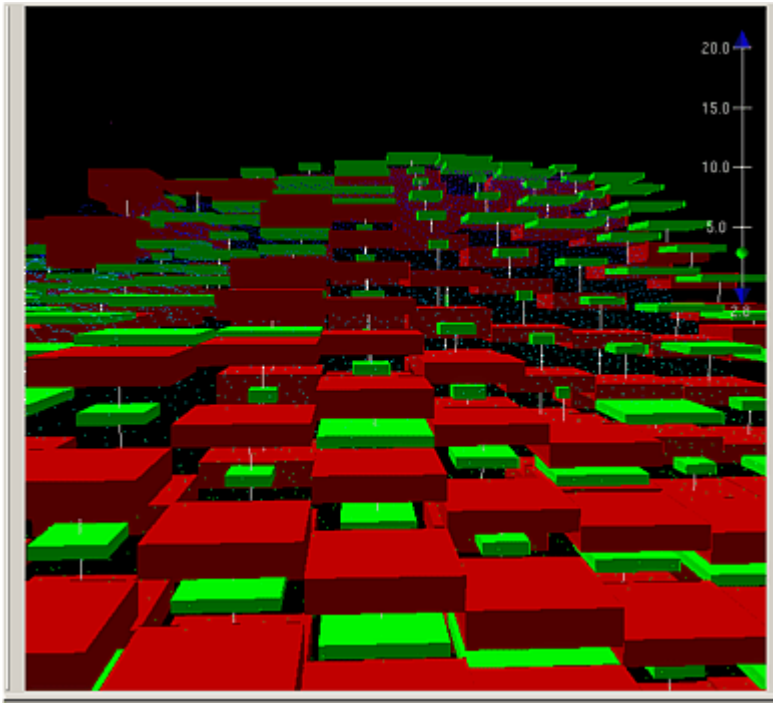
The 3-D View is refreshed to display the squares as flattened and in a standard size.

Nominate Alternative Hypotheses

Some nodes have alternative hypotheses that were not selected during disambiguation You can view alternative hypotheses and compare them against the selected ones. If needed, the alternative hypotheses can be nominated to replace the established hypotheses.

9. Select the *Show Alternative Hypotheses* check box and click **Apply**.

The 3-D and 2-D Views are refreshed to display the alternative hypothesis. The following graphic shows the 3-D View with the alternative hypotheses displayed.



The red squares represent hypotheses that were not selected during disambiguation. The vertical lines that run between the nodes are viewing aides to match the alternative hypotheses to the selected ones.

1. To replace an hypothesis, select the alternative hypothesis.

The alternative hypothesis is highlighted. The Nominate and Clear commands are now active.

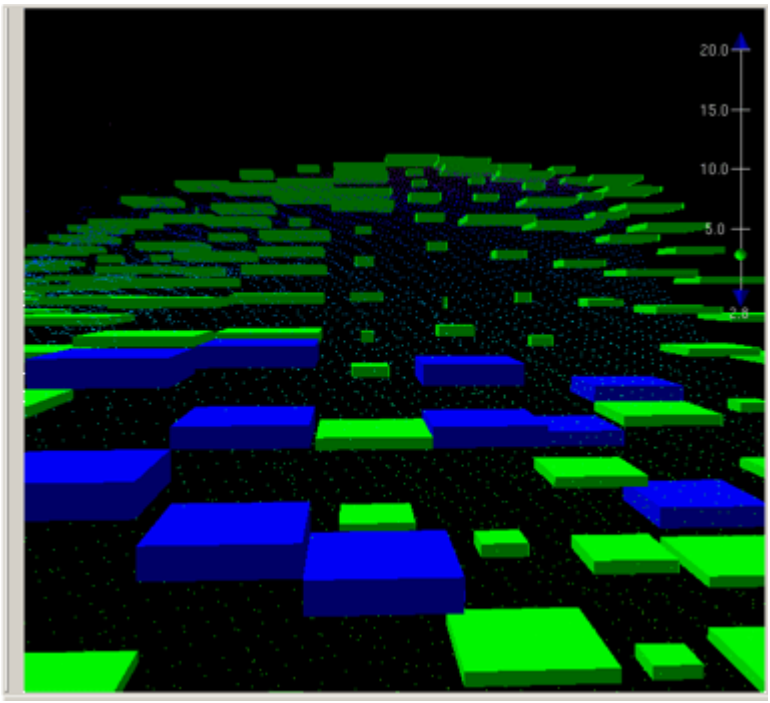
2. Select the Nominate command.

Menu	Tools > Subset Editor > Nominate
Tool	

The alternative hypothesis is highlighted in blue. This means that the nominated hypothesis has replaced the established one. All nominated hypotheses are given the highest confidence value of 0.0.

3. To compare nominated nodes against the existing nodes, clear the *Show Alternative Hypotheses* check box and select the *Show Only Selected Alternatives* check box.

The Views are refreshed to show only the nominated and established hypotheses. The following image shows the 3-D View with only nominated and established hypotheses displayed.



4. To remove the nomination flag, select the Clear command.

Menu	Tools > Subset Editor > Clear
Tool	

5. When you have completed nominating alternative hypothesis data, select the save command.

Hypotheses Cleaning

Like soundings in the Swath and Subset Editors, hypotheses can be given a rejected flag. When a hypothesis is rejected, it is retained in the Surface but is excluded from processing when disambiguation is run again or when filtering is applied.

Reject

1. Select a hypothesis
2. Select the Reject command.

Menu	Tools > Subset Editor > Reject
Tool	

The hypothesis is now flagged as rejected.

To change rejected back to the accepted status (and therefore available for further processing).

Accept

3. select a hypothesis.
4. Select the command.

Menu	Tools > Subset Editor > Accept
Tool	

The hypothesis is now flagged as accepted and is available for further processing.

Reject Node

To reject all hypotheses at a node:

1. Select a hypothesis.
2. Select the command.

Menu	Tools > Subset Editor > Reject Node
Tool	

All hypotheses associated with the node are rejected.

Accept Node

To return hypotheses associated with a node to their normal status.

1. Select a hypothesis
2. Select the command.

Menu	Tools > Subset Editor > Accept
Tool	

All hypotheses associated with the node are flagged as accepted.

Update CUBE Surface

If changes are made, the CUBE Surface can be rebuilt to show these changes using the following methods:

- Select the Recompute BASE Surface command (see "RECOMPUTE" ON PAGE 423
- Select the *Automatic BASE Surface Update* check box in the Options dialog box (see "GENERAL" ON PAGE 75)

19

Statistical Surface Cleaning

Overview

The surface cleaning function is used to perform area-based statistical data cleaning on multibeam data in HIPS.

The process uses advanced tiling technology for subdividing the entire area into variable size and density cells. It then applies rigorous polynomial regression statistics to compute the surface.

Tiling

In surface cleaning, tiling is used to subdivide the area into manageable cells where each tile is considered separately during the polynomial regression and statistical threshold testing. The tile split criteria is based either on the density of soundings within the tile or the tile size, or both.

The Surface Cleaning function automatically creates a Starting Level Tile set (even if you do not select the tiling option). The Starting Level Tile Set is created by taking the largest dimension of the field sheet and dividing the area by 3 (to the nearest power of two, starting at 0.02). If your field sheet is 75x93, take 93/3=31, and the nearest tile size is 20.48 (look for the nearest tile rounding down).

Surface Cleaning

For each tile, a polynomial solution is computed using iterative least-squares re-weighted regression. The objective of the regression is to generate a surface defined by a set of polynomial terms, which closely matches most of the real depths. However, since the number of terms defining the polynomial surface are less than the number of points considered, then the points do not fit the surface exactly. Using the residual values of each point as a goodness measure, outliers can be detected by testing for high residual values.

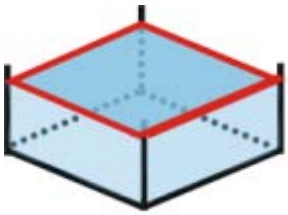
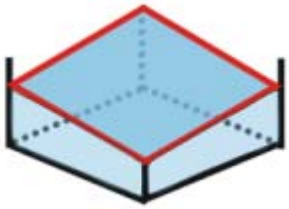
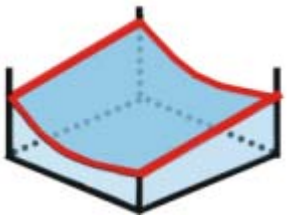
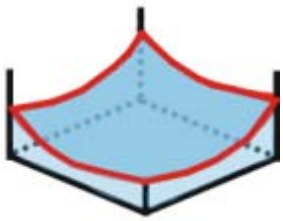
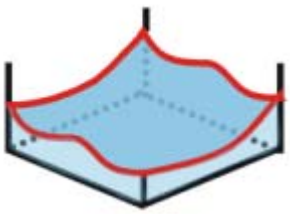
The process is iterative within each tile, and for each new iteration the soundings are re-weighted inversely proportional to their previous residual. You can fix the re-weighting scheme to apply full weight to all soundings within a defined limit from the surface. This re-weighting limit can be based on a fixed distance or a multiple of the standard deviation of the residuals within the tile.

You can also define the maximum number of iterations. The buffer around each tile determines how much extra data is used to compute the tile polynomial surface. This helps eliminate edge effects during statistical testing.

The second stage of surface cleaning is the testing of the residuals once the surface has been computed. The relevant parameters are the rejection threshold scaling values in both the positive and negative directions (above and below the surface), and the minimum absolute residual of an outlier. The latter parameter specifies that no point within this distance (either above the surface or below) are rejected. This is an important parameter if the local standard deviation of a tile is small, because the scaling values are multiplied by a small standard deviation and may reject points which are only slightly removed from the surface. This parameter is location dependent and maybe used, for example, to prevent the rejection of rocks of a specific height. The rejection threshold values are also location dependent and surface dependent.

The following table lists each of the polynomial surface types with the corresponding equation and a graphical example of each surface type.

Polynomial Surface Types

Polynomial Surface Types		
Surface	**Equation**	**Example**
Mean	$-f(xy) = b0$	
Tilted Plane	$-f(xy) = b0 + b1x + b2y$	
Curved Tilted Plane	$-f(xy) = b0 + b1x + b2x + b3xy$	
Quadratic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2$	
Cubic	$f(xy) = b0 + b1x + b2x + b3xy + b4x2 + b5y2 + b6x2y + b7xy2 + b8x3 + b9y3$	

Creating Tiles and Cleaning Data

Data must be merged before surface cleaning can be performed.

1. Select a field sheet.
2. Select the Surface Cleaning command.

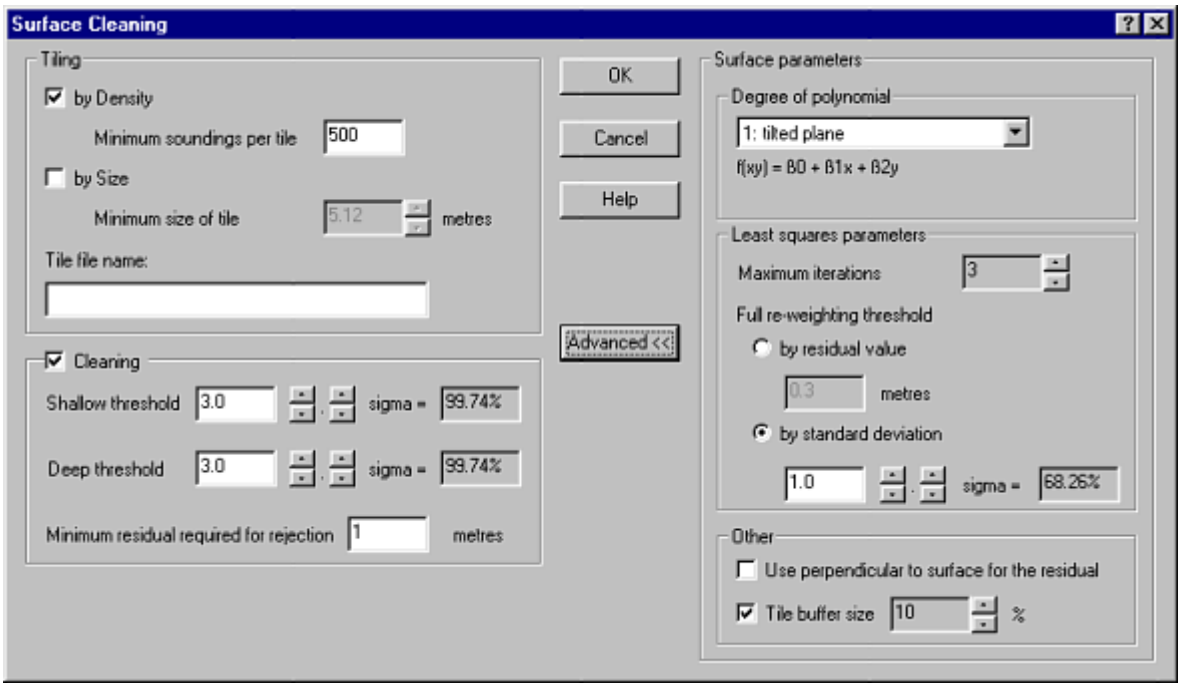
Menu	Tools > Surface Cleaning
Tool	
Pop-up	Surface Cleaning

The Surface Cleaning dialog box is displayed.

3. Enter tiling parameters, as needed, in the following fields:
   - *By Density*: When this box is checked, each tile is continuously sub-divided until one of the sub-tiles has less than the minimum number of soundings.
   - *Minimum soundings per tile*: The minimum number of soundings in a tile.
   - *By Size*: When this check box is selected, each tile is continuously sub-divided until the sub-tiles reach the minimum size (this size cannot be larger than the Starting-Level Tile Set.
   - *Minimum size of tile*: The minimum size for a tile.
4. Type a name for the tile file. All tile files are assigned an .mts extension. Also created is a .mmi file that records the list of track lines used to create the tiles.
5. Select the *Cleaning* check box to enable the surface cleaning functions.

6. Enter cleaning parameters, as needed, in the following fields:
   - *Shallow threshold*: The threshold above the polynomial surface. The value is selected by clicking the up and down arrow buttons beside shallow threshold field. The sigma value is a multiple of the standard deviations of residuals in the tile.
   - *Deep threshold*: The threshold below the polynomial surface. The value is selected by clicking the up and down arrow buttons beside deep threshold field. The sigma value is a multiple of the standard deviations of residuals in the tile.
   - *Sigma*: The equivalent confidence values are displayed as a percentage.
   - *Minimum residual required for rejection*: A sounding that fails the threshold test will not be rejected if the absolute value of the residual is less than the minimum.
7. Click **Advanced** to view more cleaning options.

The Surface Cleaning dialog expands to display more cleaning options.



8. Enter advanced tiling parameters, as needed.
   - *Degree of polynomial*: Selection of available polynomials. Select the type that matches the expected complexity of the sea floor.
   - *Maximum iterations*: The number of times the least squares computation of the polynomial processes the soundings. The computing continues until there are no additional rejected soundings or the maximum number of iterations is reached.
   - *By residual surface*: The full re-weighting of soundings for each iteration can be fixed as a distance from the surface.
   - *By standard deviation*: The full re-weighting of soundings for each iteration can be a multiple of the standard deviation of the residuals.
   - *Perpendicular to surface*: When this option is checked, the residual is calculated as the perpendicular distance between the sounding and the nearest point on the polynomial surface. The default calculation for the residual uses the depth difference between the sounding and the polynomial surface at the location of the sounding.
   - *Tile buffer*: The extra area around the tile that is included in the surface computation. The value is a percentage of the tile size and describes the width of the buffer around the tile.
9. Click **OK**.

New tiles are stored in the file that was selected in Step 6.

Query Options

You can use two query commands to view attributes of tile data.

- **Query Tile Surface**: Display the polynomial surface parameters of selected tiles.
- **Query Tile Soundings**: Display information on all the soundings plus the residuals of selected tiles.

Other tile query and display options are discussed in the section on field sheets (see "TILING" ON PAGE 355).

Query Tile Surface

1. Make sure the tiles are open in the Display window.
2. Select the tile layer in the Session tab of the Control window.
3. Press and hold the mouse button while dragging the cursor over the tiles to be queried.
4. Select the Query Tile Surface command.

Menu	Edit > Query Tile Surface
Tool	

The tile surface information is displayed in the Worksheet window.

- *Count*: number of points in the tile
- *Rejected*: the number of points rejected during surface cleaning.
- *Z Mean*: the mean depth in the tile.
- *Residual_Sigma*: standard deviation for residuals.
- *Polynomial terms*: the number of terms with values depends on the degree of polynomial you selected in the Surface Cleaning dialog box.

Query Tile Soundings

1. Make sure the tiles are open in the Display window.
2. Select the tile layer in the Session tab of the Control window.
3. Press and hold the mouse button while dragging the cursor over the tiles to be queried.

The selected tiles are highlighted.

4. Select the Query Tile Soundings command.

Menu	Edit > Query Tile Soundings
Tool	

The following data is displayed in the Worksheet window.

- *East*: easting coordinate for soundings.
- *North*: northing coordinate for soundings.
- *Depth*: depth of sounding.
- *Surface Depth*: depth of polynomial surface at the location of the sounding.
- *Residual*: difference between the sounding and the polynomial surface.
- *Shallow Threshold*: depth of rejection threshold above the polynomial surface.
- *Deep Threshold*: depth of rejection threshold below the surface.
- *Status*: status of the sounding.

Tile Histogram

The Tile Histogram dialog box displays the distribution of residuals within a tile.

If the histogram resembles a bell curve then the selected polynomial is well suited for the area. However, if the histogram has many peaks or a wide centre, then the polynomial surface is not suited to the area. If there are severe shifts in data distribution, then the area may have to re-tiled using a higher degree of polynomial.

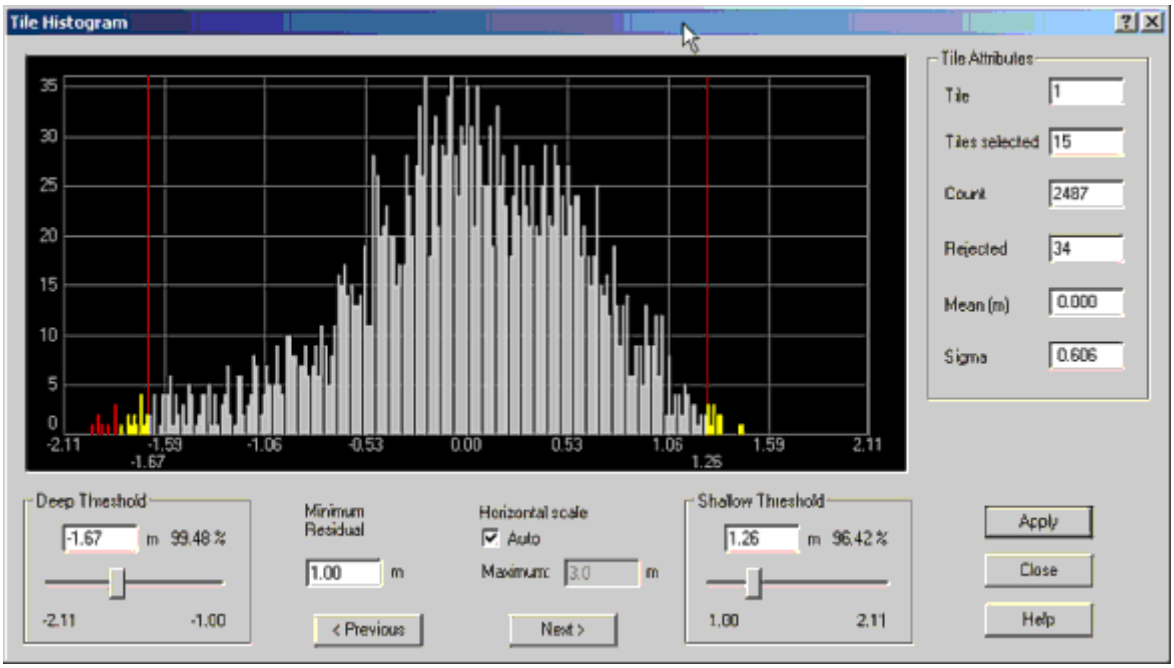
You can also use the histogram to change the threshold values and reject soundings.

Open the histogram

1. Select the tiles that you want to examine using the tile selection command.
2. Select the Tile Histogram command.

Menu	Edit > Tile Histogram
Tool	

The Histogram dialog box is displayed.



Information about the tile is displayed on the right-hand side of the dialog box.

3. To view other tile data, click **Next Tile** or **Previous Tile**

Change threshold levels

1. Position the cursor over the *Shallow Threshold* or *Positive Threshold* slide bars and while pressing the mouse button move the bars to a new position.

The corresponding confidence intervals and Maximum Residual Distance values change as the bars are moved.

2. Click **Apply**.

The rejected residuals are displayed in red.

20

Lidar Data

Overview

Light direction and ranging (Lidar) uses laser technology to measure bathymetry. In such surveys, laser wavelengths are transmitted to the water from a transceiver mounted on a plane or helicopter. Sounding data is returned to the transceiver in a continuous stream that forms a swath-like pattern.

> HIPS currently supports the Laser Airborne Depth Sounder (LADS) system and the Scanning Hydrographic Operational Airborne Lidar Survey (SHOALS) system.

Soundings are recorded in an bathymetry file. Each sounding has

- a latitude and longitude position
- a depth value
- a timestamp
- an across-track and along-track position
- a quality flag
- a status flag.

Waveform data is also included for each sounding. This data is stored in a waveform file (see "WAVEFORMS VIEW" ON PAGE 473).

Sounding Quality and Status Flags

Quality flags

When a lidar sounding is converted into HIPS, it is given a quality flag that can be changed during processing. The following table outlines the quality status that can be attached to a sounding.

Quality Flag	Description
0	The sounding has a detected depth only. The alternative depth is invalid.
1	The sounding has both a detected and a valid alternative depth. A quality control/assurance decision has not yet been made.
2	The sounding has a detected depth and a valid alternative depth, and the user has decided to keep the detected depth.
3	The sounding has both a detected and alternative value, but the alternative value has been swapped with the detected value.

Status flags

Status flags are assigned to soundings during conversion, these status are determined from information provided by the specific acquisition format.

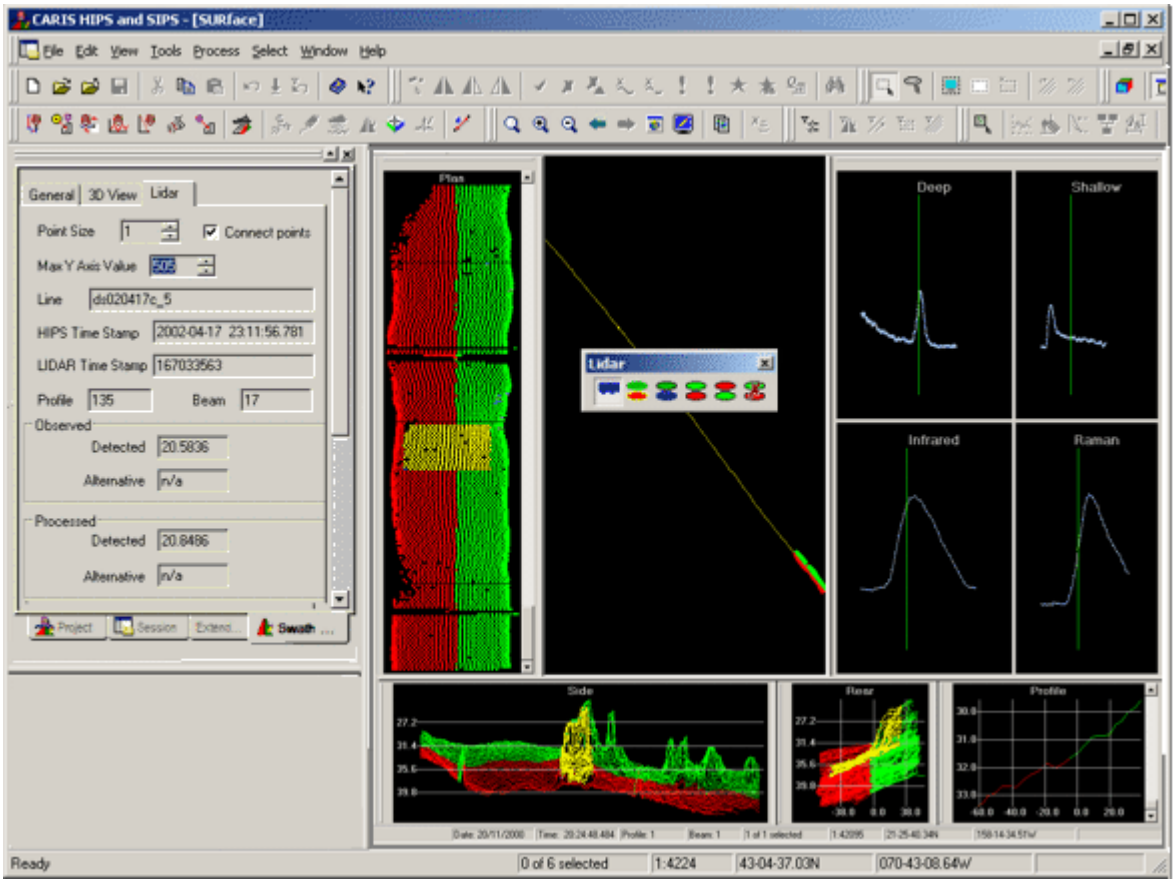
Lidar Data in HIPS

Lidar projects are organized in the same Project-Vessel-Day-Line hierarchy as sonar projects. Each project contains the following files:

- HIPS Vessel File
- an Observed Depths file (along-track and across-track sounding positions before merge)
- a Processed Depths file (lat/long sounding positions after merge)
- a bathymetry file
- a waveform file
- gyro data
- navigation data
- tide data

The default location of all the above files is ..\Hips\HDCS_Data.

To open Lidar data, see "SELECTING DATA" ON PAGE 475.



Swath Editor displaying SHOALS lidar data with the Waveform view and Lidar tab visible.

Toolbar

The toolbar controls the display and status options for soundings and associated waveform data in the Swath and Subset Editors.

This is a floating toolbar that can moved or docked anywhere in the HIPS main window. The toolbar is not automatically displayed when the Swath Editor or Subset Editor is opened.

1. Select the Lidar Toolbar command.

Menu	View > Toolbars > Lidar
Pop-up	Lidar

Waveforms View

The Waveform view displays the waveform data for a super-selected sounding (see "SELECTING DATA" ON PAGE 475 for more information on superselection). The display in the Waveform view is determined by whether the data is in SHOALS or LADS format.

1. Select data in any of the Swath Editor or Subset Editor views.
2. Select the View Waveforms command.

Menu	Edit > Lidar > View Waveforms
Tool	

The first sounding in the selection is superselected when the Waveforms window is opened.

SHOALS

There are four panels in the view. Each panel shows a graphed view of specific waveform data for the superselected sounding. The X axis represents time and Y axis represents amplitude.

LADS

The LADS waveform window contains one panel that shows a graphed view of specific waveform data for the super selected sounding. One line represents the detected depth sounding and the other line represents the alternative depth sounding (if applicable).

To view the complete waveform for SHOALS data, the Y-axis scale may have to be increased. The default value is 255.

1. Click the *Max Y-Axis Value* up or down arrow buttons until the entire wave form is within view.

Point display

The points that describe the waveform graphs can be increased or decreased.

1. Select the Lidar tab in the Control window if it is not already displayed.
2. Click the *Point Size* up and down arrow buttons.

As the value changes, the displays in the four windows are redrawn to show the points at the selected size.

3. To draw a line through the points in the view, select the *Connect Points* check box.

Selecting Data

Open data

Lidar data can be viewed and cleaned in the Swath and Subset Editors.

1. Open either the Swath Editor or the Subset Editor.
2. Select the Lidar Toolbar command.

Menu	View > Toolbars > Lidar
Pop-up	Lidar

The Lidar toolbar is visible.

3. Select the View Waveforms command.

Menu	Edit > Lidar > View Waveforms
Tool	

The Waveforms window (see "WAVEFORMS VIEW" ON PAGE 473) is displayed and the Lidar tab is displayed in the Control window. No data is visible in the Waveforms window yet.

There are two levels of selection in Lidar mode: regular selection and superselection.

- Selection refers to all highlighted soundings in any of the Views.
- A superselected sounding is a subset of the selected soundings and is highlighted in a different colour from the regular selection. When a group of soundings are selected, the first in the selection (i.e. the lowest beam and profile number) is automatically super-selected.

All selected soundings can be queried or rejected, but you can only view waveform data and sounding attributes for a superselected sounding. Make sure that the Waveforms window is displayed. When this window is closed, the superselection option is inactive.

4. Choose an Editor Selection mode (see "QUERY LINE DATA" ON PAGE 220). The Edit By Range option is selected by default.
5. Select data in any of the views.

The data is highlighted in the selection colour determined in the Options dialog box (see "LIDAR" ON PAGE 83). The superselected sounding is displayed in the Plan View (Swath Editor) and the 2-D and 3-D Views (Subset Editor).

The waveform data for the superselected sounding is displayed in the Waveforms window.

The following attributes for superselected sounding are displayed in the Lidar tab in the Control window:

- *HIPS Time Stamp*: Year-Month-Day-Hour-Minute-Seconds-Milliseconds.

- *LIDAR Time Stamp*: Half milliseconds since midnight (SHOALS only).
- *Profile number*: The along-track position of the sounding
- *Beam number*: The across-track position of the sounding.
- *Detected depth*: If this depth is selected for the Observed Depth file (depths prior to merge), then an asterisk is displayed beside the field.
- *Alternative depth*: If this depth is selected for the Observed depth file (depths prior to merge), then an asterisk is displayed beside the field.
- *Processed depth*: The depth value for the sounding after the Merge process
- *Confidence value*: The confidence value for the sounding (SHOALS only).
- *Tide*: Tide data for the sounding. Data must be merged before this value can be displayed.
- *Status*: Current status flag for the sounding.

You can use the Lidar tab to superselect a neighbouring sounding.

6. To view another sounding in the current selection, click the **Next** or **Previous** button in the Lidar tab of the Control window. This moves you to the next or previous selected sounding by ping and beam number.

The graphs in the Waveforms dialog box is refreshed to show the new data and the new superselected sounding's attributes are shown Lidar tab data fields.

Find sounding

You can also super select soundings by locating it by the time stamp (SHOALS data only).

1. Type the Lidar time stamp in the *Time Stamp* field.
2. Click **Find**.

The sounding with that time stamp is highlighted as super selected and the sounding data is displayed in the Lidar tab fields.

Super select data in the Worksheet window

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

Queried data can also be superselected.

1. Select data in any of the views.
2. Select the Query command.

The selected data is displayed in the Query tab of the Worksheet window.

3. To superselect a sounding, click a row so it is highlighted.

The beam is highlighted as super selected in the Plan View.

Selecting Detected and Alternative Data

In some instances, two depth values (detected and alternative) are returned. You can decide which depth value is most relevant and change the sounding's quality flag accordingly (see "SOUNDING QUALITY AND STATUS FLAGS" ON PAGE 470).

Depth values can be selected and changed in both the Swath Editor and the Subset Editor.

Highlight alternative soundings

Menu	Edit > Lidar > Highlight Alternative Depths
Tool	

This command highlights soundings that contain valid alternative depths. It does not select or superselect soundings.

1. Select the Highlight Alternative Depths command. This command highlights only data in the Plan View.

Soundings with a Quality flag of 1, 2, or 3 are highlighted in colour that is selected in the Options dialog box (see "LIDAR" ON PAGE 83).

You can select from three options.

View all alternative depths

Menu	Edit > Lidar > View Alternative Depths
Tool	

This command refreshes the display so that the alternative depths for soundings with a Quality 1 flag are shown. This is a useful tool for viewing the position of alternative depths in relation to neighbouring soundings.

1. Select the View Alternative Depths command.

The data in the Views refresh to display the alternative soundings.

Swap to alternative

This command applies an alternative depth in a superselected sounding to the detected depth. With SHOALS data, the depth is swapped; and with LADS data, the depth and position are swapped.

1. Select soundings and super select a sounding with a Quality 1 flag.
2. Select the Swap to Alternative command.

Menu	Edit > Lidar > Swap to Alternative
Tool	

An asterisk beside the depth value indicates whether the detected or alternative depth is currently in the HIPS depth file.

Keep detected

This command is used to verify that the detected depth is preferred over the alternative depth. The Quality flag is changed to 3. (see "SOUNDING QUALITY AND STATUS FLAGS" ON PAGE 470).

1. Select soundings and super select a sounding with a Quality 3 flag.
2. Select the Keep Detected command.

Menu	Edit > Lidar > Keep Detected
Tool	

Cleaning Sounding Data

Reject Data

Soundings can be assigned an Rejected by Disabling Sounding flag. These soundings are not included in the merge or any post-processing operations. Data can rejected in the Swath and Subset Editors.

1. Select a sounding or group of soundings.
2. Select the Reject by Disabling Sounding command.

Menu	Edit > Lidar > Reject by Disabling Sounding
Tool	

The soundings are now flagged as rejected.

Accept data

You can return soundings to the accepted status.

1. Select the Show Rejected command so the disabled soundings are visible.
2. Select the soundings.

Menu	View > Show Rejected
Tool	
Key	<F4>

3. Select the Accept command.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

21

Calibration

Overview

In HIPS and SIPS 6.1, the Calibration module still resides within the Subset mode of the HDCS program from past releases of HIPS. Calibration is performed by visually comparing the effects of various sensor offsets on calibration survey lines. The initial parameters are extracted from the HIPS Vessel File (HVF) and modifications can be written to the same HVF.

Calibration surveys are performed to allow adjustments of the survey data for parameters such as transducer mounting offsets (pitch, roll, yaw), and navigation latency. Other parameters, such as heave latency, can also be adjusted.

Calibration lines

Calibration lines are a set of survey lines specifically designed to allow for calculation of minor sensor deviations from established parameters. By running the vessel over known sea bottom topography, at different speeds and directions, correction factors can be derived for various devices on the ship. These corrections are then used for the duration of the survey mission to ensure correct processing of data.

Workflow

In HIPS and SIPS, Calibration is a six-stage process.

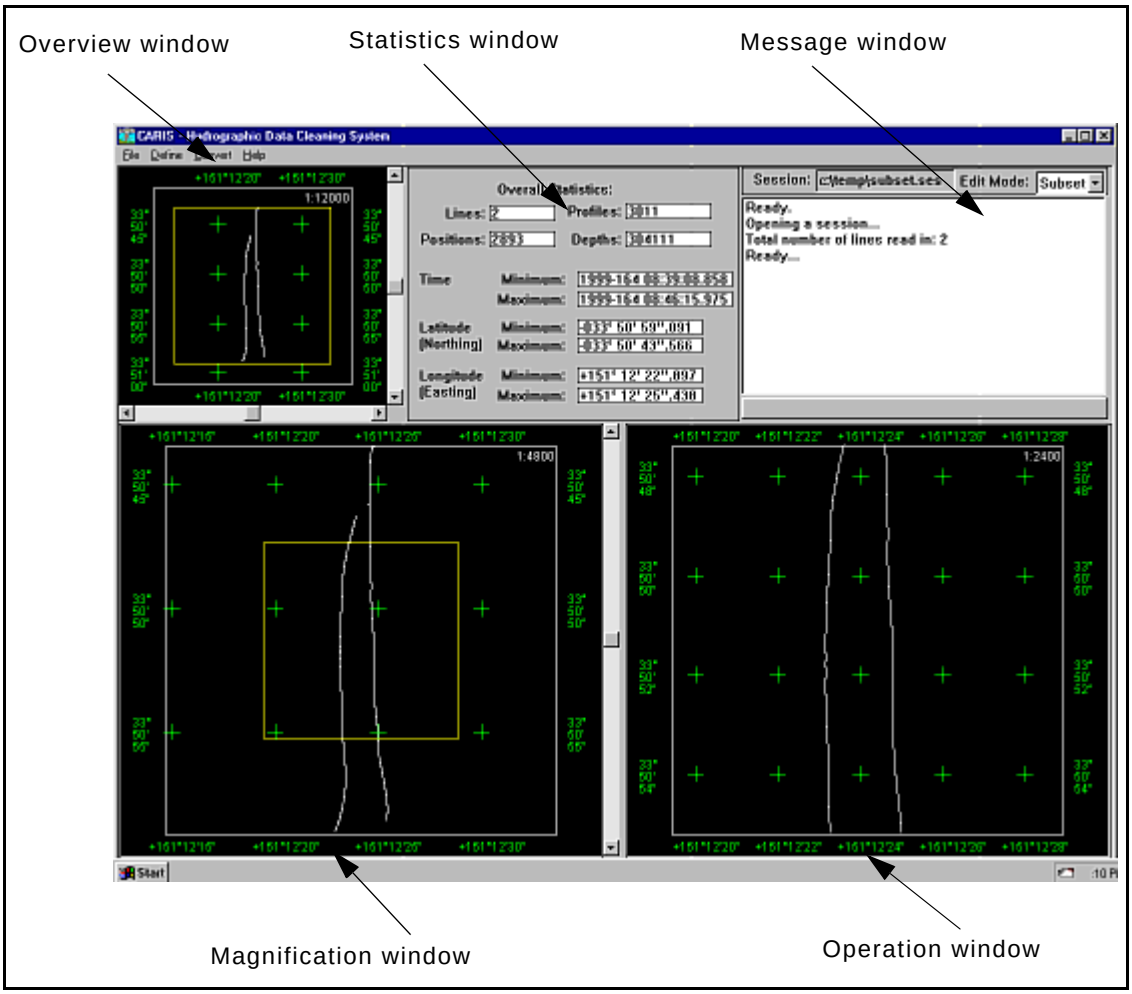
1. Open the calibration lines in HIPS and SIPS.
2. Select a pair of lines that were surveyed for a specific parameter.
3. Use the Calibration function to launch the Hydrographic Data Cleaning System (HDCS) directly into the Subset mode.
4. Create the subset in HDCS.
5. Use the Calibration tool in HDCS to measure and view the effects of the adjustments.
6. Make a note of the final adjustment values or save them directly into the HVF.

Create Subsets in HDCS

1. In HIPS and SIPS, select the track line(s) you want to calibrate.
2. Select the Calibration command.

Menu	Tools > Calibration

The HDCS interface is opened in Subset mode.



The track lines are shown in the HDCS interface. The Magnification window controls the track line display in the Operation window

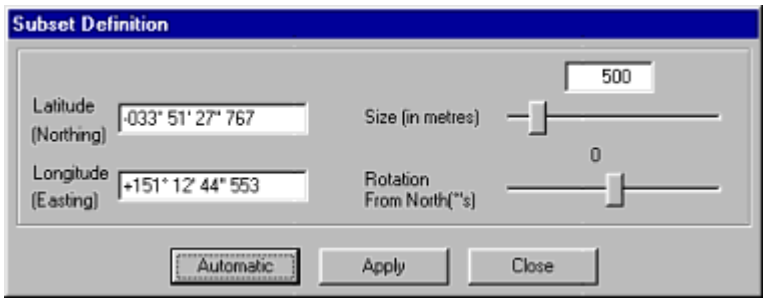
3. Position the cursor in the Magnification window.
4. Press and hold the mouse button while dragging the bounding box corners to the edge of the window so all the track line area is displayed in the Operation window.

Define a subset

Pop-up	Define Subset
Key	<Insert>

5. In the Operation window, right-click the mouse and select the Define Subset command from the pop-up menu.

The Subset Definition dialog box is displayed and a subset area bounding box is opened in the Operation window.



6. To determine the subset area, position the cursor on the *Size* slider control; press and hold the mouse button and move the slider to the desired position.

As the slider is moved, the subset area is resized.

7. To determine the subset angle, position the cursor on the *Rotation From North* slider control; press and hold the mouse button and move the slider to the desired angle.

As the slider is moved, the subset is rotated.

8. Register the subset by positioning the cursor in the subset area of the Operation window and right-clicking the mouse button.

The subset area is highlighted in blue and automatically numbered.

9. If you are creating more than one subset area, position the cursor on the subset; press and hold the mouse button and drag the subset bounding box to an adjacent survey area (make sure there is some overlap between the two areas).
10. Repeat Steps 8 and 9 until all the survey area is registered as subsets.

Clean subset

Data must be displayed in the spatial subset mode before calibration can begin.

1. Click a subset so it is highlighted in yellow.
2. Select the Clean Subset command.

Pop-up	Clean Subset
Key	<F2>

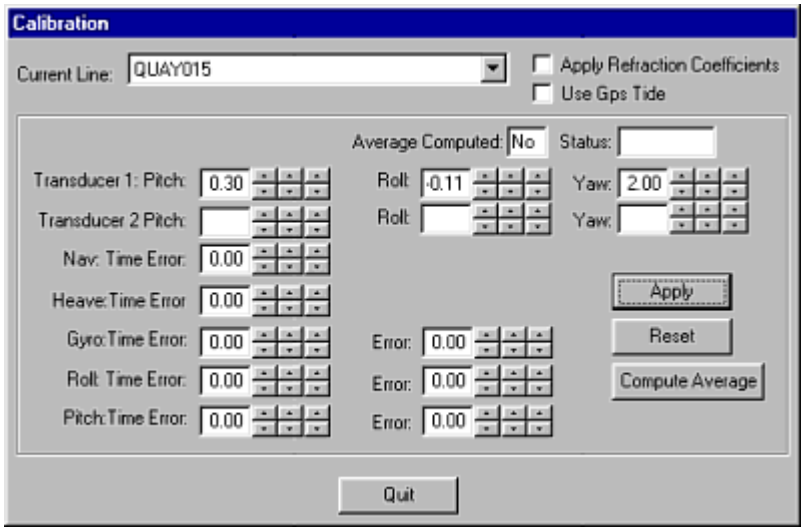
The soundings are now displayed in spatial subset mode.

Perform Calibration

Pop-up	Calibration

1. Right-click in the Operation window and select the Calibration command.

The Calibration dialog box is displayed.



When Calibration mode is in effect, the display in the Operations window reflects the locations of soundings as if they were merged using the on-screen parameters.

In the Operations window, you retain the ability to switch view points, pixel sizes, and depth range, but not the ability to select soundings for query or rejection.

The Magnification display is not updated with calibration-adjusted soundings until the parameters are saved to the HVF, and the lines are re-merged. You can still generate mean and standard deviation maps, but only with unadjusted soundings. When Calibration mode is started, the current values from the HVF are accessed and displayed.

2. Select any of the following options:
   - *Current Line*: Select the line to apply calibration adjustments. Usually all calibration lines are from the same vessel (and, therefore, the same HVF). However, you are not restricted to using lines with the same HVF. Other lines can also be opened in dialog box. If the parameters with that line are associated with other lines then those lines are updated too.
   - *Apply Refraction Coefficients*: Select this option to apply refraction coefficients. The effects of the Refraction Editor data on the final depths can be changed at any time by editing the coefficients and re-merging. The effects can removed by re-merging and choosing not to apply the coefficients.

- *Apply GPS Tide*: Select this option to apply GPS tide. The effects of GPS tide data on the final depths can be changed at any time by editing the Sounding Datum Height and re-merging. The effects can be removed by re-merging and choosing not to apply the GPS Tide.
- *Up/Down Arrow Buttons*: Use these buttons to increment/decrement parameter values. The first row of buttons adjusts the integer portion of the value; the second column adjusts in 0.1 increments, and the third column adjusts in 0.01 increments.

3. Once data has been selected or modified, you can select from among the following options:
   - *Apply*: Effect all changes immediately.
   - *Reset*: Reset all HVF parameters to their original settings.
   - *Compute Average*: Append the settings to an internal buffer to compute the average settings. This is done because one set of parameters may work well in one area of a survey, but not in another area of the same survey. You can then have a set of averaged settings for the entire line.
     - When calibration mode is started, the buffer is empty. Values extracted from the HVF are *not* put into it.
     - If selected without any changes being made, the buffer is not updated (i.e., it remains empty).
     - The first value entered (not the value extracted from the HVF) is the first value put into the buffer (after Compute Average is selected), and subsequent values is averaged with it.
     - You are prompted to save settings in the HVF only if you submitted values for averaging. Even if only one number was entered, you must select Compute Average if you want the setting to be saved in the HVF when you exit.
   - **Quit**: Submit the values to the buffer for averaging; or close the calibration session. A calibration session remains in effect until **Quit** is selected. You cannot leave Subset mode while in Calibration mode.

If **Quit** is selected, a dialog box prompts you to update the HVF with the modified values.

4. Click **Yes**, to confirm the modification.

When you exit the Calibration mode, you must merge the line(s) again for the calibration adjustments to take effect (see "MERGE" ON PAGE 272).

Sample Calibration Procedure

The following are sample calibration procedures[1] for adjusting sensor offsets. In this procedure Navigation Time Error, transducer Pitch offset, transducer Roll offset, and transducer Azimuth (or Yaw) offset are adjusted. Initially, all offsets are set to zero (in the HVF).

Take care in cleaning position and attitude data. Depth cleaning is not as vital because you are looking at data trends and not individual spikes. Navigation Time Error, Pitch offset, Roll offset, and Azimuth (Yaw) offset should be set to zero in the HVF. Merge all lines.

Navigation time error

The Navigation Time Error adjustment is performed on sets of two coincident lines, run at different velocities, over sloping terrain or a conspicuous topographic feature. You must view a profile of the two centre beams.

1. In Subset Edit determine the along-track horizontal displacement between the two centre beam profiles of one set of coincident lines. Repeat this at several locations.
2. Compute the mean displacement.
3. Determine the Navigation Time Error as displacement / (high speed – slow speed).
4. Enter Calibration mode and enter the mean value (select **Apply** if necessary).
5. View result of correction, if it is not to your satisfaction, use the arrow buttons to adjust the value.
6. Select **Compute Average**.
7. Move to another location within the subset and repeat Steps 5 and 6.
8. Once the subset is to your satisfaction quit the calibration, confirm the update of the HVF and merging of all lines.
9. If calibration data covers more than one subset, repeat Steps 1 through 8 for all other subsets. Be sure to write down the final correction from each subset.
10. Manually average corrections from all subsets
11. Update the HVF.
12. Re-merge all lines (see “MERGE” ON PAGE 272).

1. These procedures are taken from the document “Field Procedures for the Calibration of Multibeam Echo-sounding Systems”, by André Godin (available from CARIS).

Transducer pitch offset

This adjustment is performed on sets of two coincident lines, run at the same velocity, over sloping terrain or a conspicuous object, in opposite directions. Of the above mentioned offsets, only the Navigation Time Error should be set in the HVF. You must view a profile of the centre beams.

1. In Subset Edit determine the along-track horizontal displacement between the two centre beam profiles of one set of coincident lines. Note the depth.
2. Determine the Pitch offset as arctan ((displacement / 2) / depth).
3. Enter Calibration mode and input the offset (select **Apply** if necessary).
4. View result of correction, if it is not to your satisfaction use the arrows to adjust the value.
5. Select **Compute Average**.
6. Move to another location within the subset and repeat Steps 4 and 5.
7. Once the subset is to your satisfaction quit the calibration, update the HVF and merge of all lines.
8. If calibration data covers more than one subset, repeat Steps 1 through 7 for all other subsets. Be sure to write down the final correction from each subset.
9. Manually average corrections from all subsets.
10. Update the HVF.
11. Re-merge all lines.

Transducer azimuth (yaw) offset

This adjustment is performed on sets of two lines, run over a conspicuous topographic feature. Lines should be run in opposite directions, with the same outer beams crossing the feature. Of the above mentioned offsets, only the Navigation Time Error and Pitch offset should be set in the HVF. You must view a profile of the same outer beams.

1. In Subset Edit find the desired topographic feature.
2. Observe an along-track profile that shows the same outer beams (from both passes) and the topographic feature.
3. Determine the along-track horizontal displacement between the two representations of the topographic feature.
4. Change bounding box orientation and view perspective in order to measure the across track distance for the observed outer beams.
5. Determine the Azimuth offset as arcsin ((displacement / 2) / across-track).
6. Enter Calibration mode and input the offset (select **Apply** if necessary).
7. View result of correction, if it is not to your satisfaction use the arrow buttons to adjust the value.
8. Select **Compute Average**.

9. Move to another set of lines within the subset and repeat Steps 7 and 8.
10. Once the subset is to your satisfaction quit the calibration, update the HVF and merge all lines.
11. If calibration data covers more than one subset, repeat Steps 1 through 10 for all other subsets. Be sure to write down the final correction from each subset.
12. Manually average corrections from all subsets.
13. Update the HVF.
14. Re-merge all lines.

Transducer roll offset

This adjustment is performed on sets of two coincident lines, run over flat terrain, in the opposite direction. Of the above mentioned offsets, the Navigation Time Error, Pitch offset, and Azimuth offset should be set in the HVF. You must view a profile of individual swaths.

1. In Subset Edit view the across track profile of the desired lines.
2. Enter Calibration mode and adjust the Roll offset until the two profiles are coincident.
3. Select **Compute Average**.
4. Move to another location within the subset and adjust the Roll off-set until the lies are coincident.
5. Select **Compute Average**.
6. Once the subset is to your satisfaction quit the calibration, agree to update the HVF and merge all lines.
7. If calibration data covers more than one subset, repeat Steps 1 through 4 for all other subsets. Be sure to write down the final correction from each subset.
8. Manually average corrections from all subsets
9. Update the HVF
10. Re-merge all lines.

22

Side Scan Data

Overview

SIPS provides tools for viewing and processing side scan sonar and multibeam backscatter data. Sensor editors allow you to display and edit towfish heading and navigation data.

The side scan data can be slant-range corrected and used to generate a georeferenced mosaic—in GeoTIFF format—with a customized colour palette. Various tools, such as Slant Range, Beam Pattern, Angle Varying Gain and Gain Corrections, are available to further correct the imagery by removing unwanted artifacts from the data.

Preferred parameters for these corrections can be saved with the SIPS Template Wizard, so that processing can be consistently applied to some or all of the entire survey.

The Signal Display Window provides a plot of signal intensity against across-track distance (or time, in raw mode). The pre-and post-processing state of any correction can be compared using the Revert and Reapply functions.

Side scan contacts can be identified and given attributes for use in a database. These can also be loaded into a CARIS map as vector symbols and lines.

Workflow

This is an outline of the steps in a typical side scan processing session:

1. Define the configuration of the survey vessel.

   See “Define Vessel File” on page 104.

2. Define a new project:
   - Assign one or more vessels to the project.
   - Define the days for which the vessels have data.
   - Store raw measurement files in a pre-processing directory.

   See “Define Project” on page 150.

3. Load and convert raw measurement data for each day into HIPS/SIPS format. This creates a series of survey lines that are stored in a generic format that makes it possible to use the same editing tools on data received from a variety of collection systems.

   See “Generic Data Parser” on page 112.

   See “Convert Survey Data” on page 111

4. Check and edit the navigation for each line, as well other side scan sensor data such as gyro/heading, towfish depth, towfish height, and cable out/layback

   See "Navigation Editor" on page 261.

   See "Attitude Editor" on page 271.

5. Check and edit sonar records

   See "Side Scan Editor" on page 494

6. Correct sonar data
   - Perform bottom digitizing.
   - Perform slant range correction.
   - Perform beam pattern correction.
   - Perform angle-varying gain correction
   - Perform gain correction.
   - Execute Despeckle filter

   See "Processing" on page 518

7. Create and edit contacts

   See "Contacts" on page 539.

8. Create a sonar mosaic using one or more processed lines.

   See "Mosaics" on page 545.

The mosaic image is generated as an image layer within a field sheet and can also be used in other CARIS products.

Side Scan Editor

Using the Side Scan Editor, you view, edit and process sonar data in preparation for creating a mosaic.

Data can be viewed in either raw or processed mode. Processing can be applied to a single ping, a range of pings, or one or more track lines.

The following sequence reflects a typical SIPS workflow.

1. When you first start the Side Scan Editor, the ping for the selected trackline is displayed in raw mode (unless slant range corrected data is available). If you have already slant-range corrected the data, you can return to raw mode to start again, however, any correction previously made to processed data will be lost.
2. If desired, the raw data may be edited by selecting, querying, rejecting or accepting pings. Rejecting pings removes them from consideration for any mosaics to be created from this data.
3. Once pings are edited, you may need to digitize the “first return” or altitude if the sonar did not detect and record this data during acquisition. The altitude is required to perform slant range correction. If digitized altitude data does exist, it may need to be edited first.
4. With fish altitude and an average sound velocity, the raw data can be slant range corrected. Slant range correct data is shown in processed mode, where the across track measurement is horizontal distance. In this processed mode, you can edit and query pings, make height and distance measurements, and mark and edit contacts.
5. For conventional side scan data, once the data is slant range corrected it can be used to generate a mosaic. However, further processing can greatly enhance the quality of produced mosaics.
6. Use the correction tools such as Beam Pattern Correction, Angle-Varying Gain Correction and Despeckle to remove unwanted artifacts.
7. Create a mosaic in the main HIPS and SIPS interface. The side scan mosaic is stored in an image format similar to a BASE surface. A custom colour palette can be created and the image exported to other image formats with the new colours. The exported image file can be used in other CARIS products and third-party software systems.

Side Scan Editor Interface

The Side Scan Editor consists of a data display window, toolbars and the Side Scan Editor tabs in the Control window.

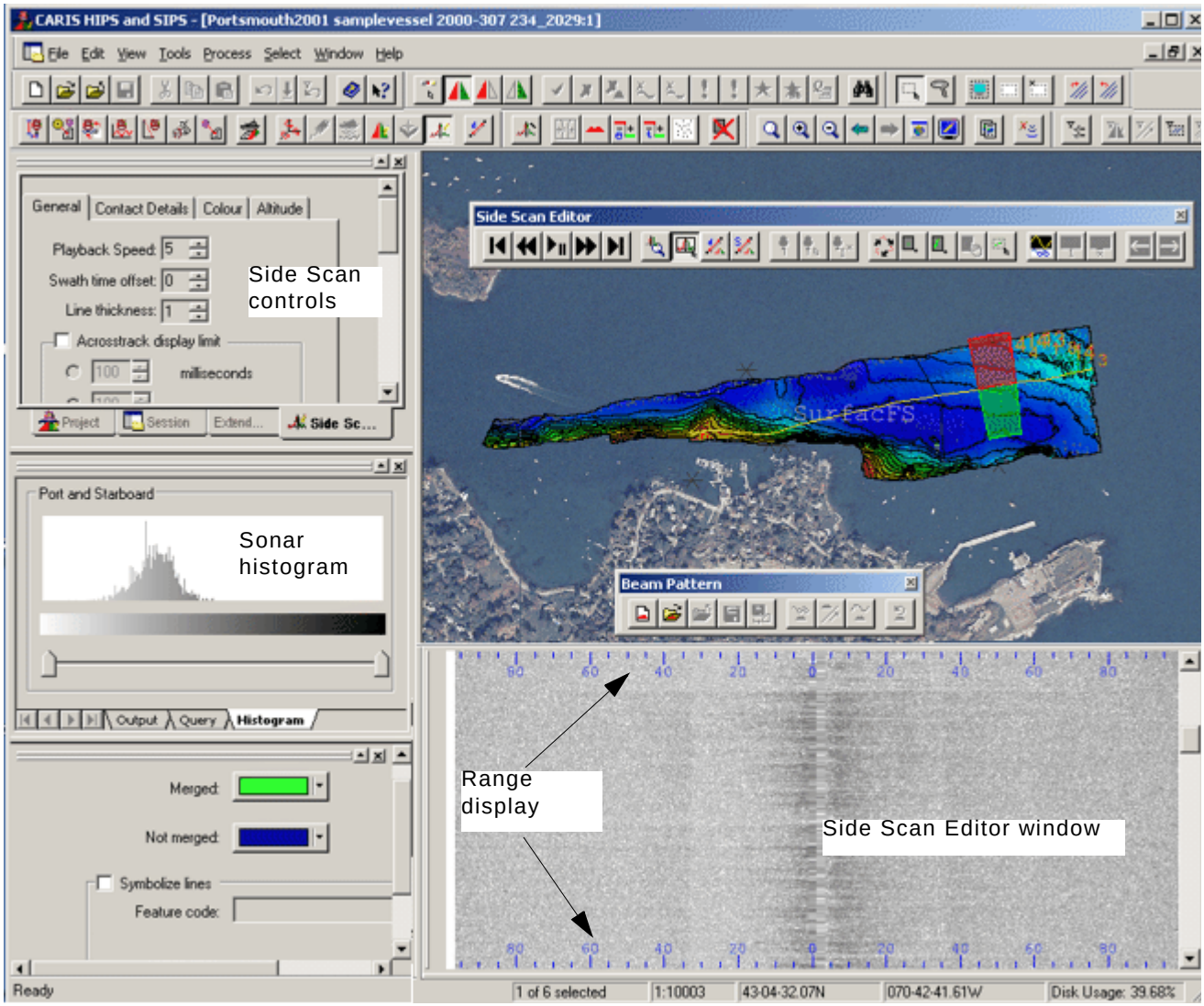
To open the Side Scan Editor:

1. Select a towfish track line in the HIPS and SIPS Display window.
2. Select the Side Scan Editor tool.

Menu	Tools > Side Scan Editor > Open
Tool	
Pop-up	Tools > Side Scan Editor

When the Side Scan Editor is opened, the data for the selected track line is displayed in the Side Scan Editor window, which has a default location along the right side of the interface.

As well, a histogram of the sonar file is displayed in the Worksheet window, and the Side Scan Editor toolbar and the Beam Correction toolbar are activated.



Side Scan Editor window

The Side Scan Editor data display window (also called the "waterfall" view) can be undocked and moved to another position, for example, at the bottom of the interface, as in the figure above (see "REPOSITION AND RESIZE WINDOWS" ON PAGE 22).

There are two modes to display data in the Side Scan Editor window:

- **Raw**: (the unprocessed sonar data). In raw mode the across-track axis represents time, thus the data is shown in the sequence in which it was received. The central portion of the image still displays the water column. This mode will always show the raw data for the selected track line.
- **Processed**: (the processed sonar data). In processed mode the across-track scale represents distance. The water column is no longer visible as it is removed during the slant range correction process. The data displayed in this mode will reflect the last processing done to it.

Menu	Tools > Side Scan Editor > Raw/ Processed
Pop-up	Raw/Processed

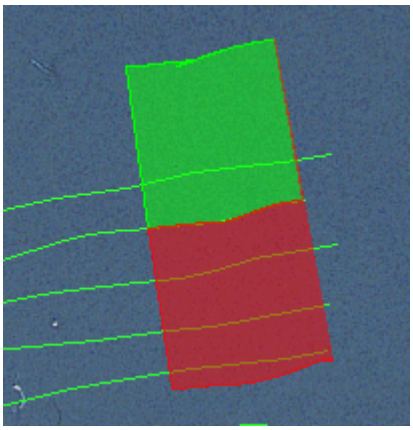
1. Select a Display mode.

The display is refreshed to show the data in the selected mode. The selected mode is indicated with a check mark in the menu.

> Note: If the data does not fill the entire height of the Editor window, it will be displayed in the bottom of the window and can be processed normally. The empty area above the data will indicate "No data to display".

Coverage outline

A coverage outline in the Display window automatically highlights the location of the side scan data that is visible in the Side Scan Editor window. The starboard and port areas of the coverage outline are appropriately colour coded. The size of the coverage outline varies with the amount of side scan visible in the Side Scan Editor window.



Colours for outlines and selection are controlled from the Tools >Options dialog box (see "SIDE SCAN EDITOR" ON PAGE 89).

Compression

The number of pixels used to display data in the Side Scan window is determined by the width of the display window and screen resolution. However, because the volume of data is often too great for the window to display all pixels, the imagery must be compressed.

There are three compression methods to choose from.

Menu	Tools > Side Scan Editor > Compression >Skip/ Average/Maximum

1. Select a Compression command .

- **Skip** selects pixels for display at regular intervals across the track.
- **Average** compiles average pixel intensity within a group of pixels spaced at regular across-track intervals.
- **Maximum** selects the brightest pixels from groups of pixels at regular across-track intervals.

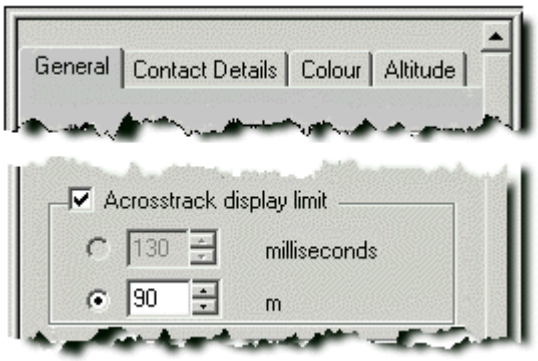
The display in the Side Scan Editor is refreshed to show the data according to the selected compression option. The default display uses the Skip option.

Side Scan Editor controls

The General tab of the Side Scan Editor contains controls for navigating and displaying data in the Editor window. This tab is visible in the Control Window when the Side Scan Editor is opened.

Across-track display

The width of the ping shown in the Side Scan Editor window can be increased or decreased by time or distance, using the controls in the *Acrosstrack Display Limit* fields.



To set across track display limits:

1. Make sure the Control window is open.
2. Select the General tab from the Side Scan Editor tab.
3. To have the entire width of the ping is shown in the Editor, leave the *Across Track Display Limit* box unchecked.
4. To limit the width of ping displayed, check the *Across Track Display Limit* box and select which value to use as limiter:
   - *Milliseconds*: Click the arrow button to change the time value. This option is automatically selected.
   - *Metres/Feet*: Click the arrow buttons to change the distance value.

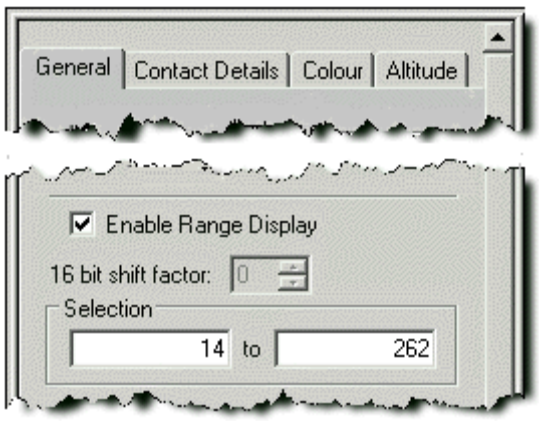
As time or distance is incremented or decreased, the amount of the data is displayed in the window is changed.

The units of distance units can be changed using the *Display* tab in *Options* on the *Tools* menu.

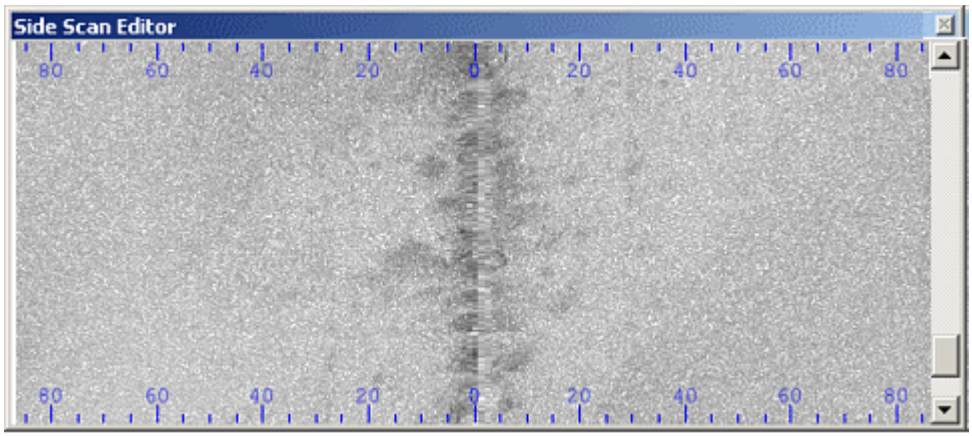
Enable Range display

The range display runs across the top and bottom of the Side Scan Editor window. In Raw mode the range displays time intervals in milliseconds. In Processed mode, distance is displayed in the units you have set in the Tools >Options >Display dialog box.

1. Select the Enable Range Display check box in the Side Scan Editor General tab.



The colour of the range display is set to the same colour used for measure/offset tools in the Side Scan Display Options of the Tools menu.



16-Bit Shift Factor

An 8-bit colour display in a 16-bit processed data set can be enhanced by bit-shifting to select which 8 bits are used to generate the display. This can create a more even distribution of pixels.

1. Make sure that 16-bit processed data is displayed in the Side Scan Editor.
2. Click the up or down arrow keys beside the 16 Bit Shift Factor field to select a value from 0 to 8.

The display is refreshed.

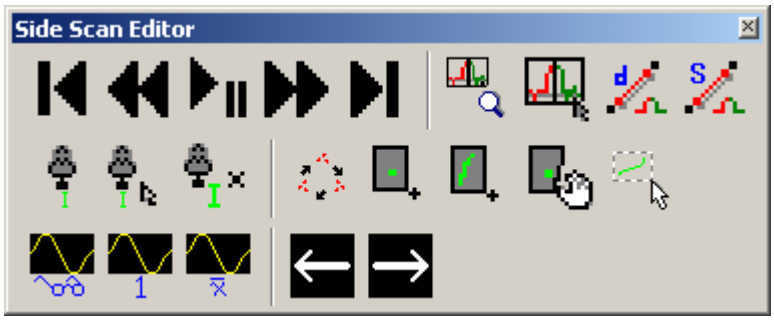
Ping selection

The two Selection fields display the start and end ping numbers when you select part of the trackline displayed in the Editor window. You can use these fields to select specific start and end pings.

Toolbar

The Side Scan Editor toolbar is displayed in the interface when the editor is started.

The toolbar can docked or undocked, or removed from the interface like other toolbars. Below is an example of the toolbar in the undocked position.



The toolbar contains controls to navigate through the track line and perform other functions.

Toolbar Buttons	Purpose
	Playback (see "Navigate in Side Scan Editor" on page 501)
	Zoom (see "Zoom" on page 503)
	Select (see "Selection" on page 504)
	Measure distance and shadow (see "Measure Distance" on page 509 and "Measure Shadow" on page 510)
	Digitize (see "Digitizing Sonar Altitude" on page 521)
	Contacts (see "Create Contacts" on page 539)
	View waveform (see "Signal Display Window" on page 515)
	Revert and reapply (see "Edit Side Scan Data" on page 510

Navigate in Side Scan Editor

You can navigate up or down the track line by clicking along the scroll bar to the right of the Side Scan Editor window.

- Clicking the scroll arrow buttons moves the view in the editor forward or backward by a single ping.
- Clicking the scroll bar, or dragging the scroll box forward or backward increments your view by a set of pings.

Alternately, you can move up or down the track line by turning the mouse wheel (if available).

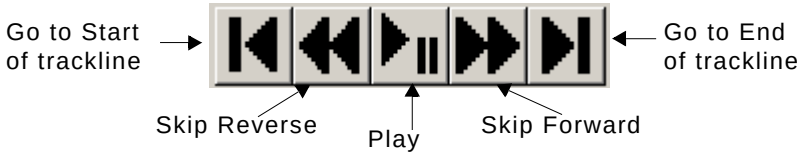
You can also navigate up or down the track line using the keyboard.

- Pressing <Page Up> or the space bar moves the editor forward by one “page”(the height (# of pings) visible in your Side Scan Editor window is the increment).
- Pressing <Page Down> or holding the <Ctrl> key while pressing the space bar moves the editor back by one “page”.
- Pressing <Home> takes you to the start of the track line, and pressing <End> takes you to the end of the track line.

Playback

The Side Scan Editor can automatically scroll data from the start of the track line to the end by using the *Playback* option.

The view in the Side Scan Editor will scroll at the speed set on the Side Scan tab of the Control window.



The ping coverage outline in the Display window moves with the display in the Editor.

Menu	Tools > Side Scan Editor > Playback Controls>Start
Tool	

1. Click Start to go to the start of the track line.
2. Click Play to start scrolling the data.

Menu	Tools > Side Scan Editor > Playback Controls> Play
Tool	

The Side Scan Editor moves up the track line ping by ping until it reaches the end of the surveyed data.

3. Reset the Side Scan Editor view to the previous position by clicking Start.

Playback speed

You can adjust the scrolling speed for automatic scrolling in the *Playback Speed* field.

1. Make sure the Control window is open and Side Scan Editor tab is displayed.
2. Select speed value from the *Playback Speed* field in the General tab by clicking the up or down arrow buttons.

You can also view the data by moving up or down the track line in larger increments: "pages" the height visible in your Side Scan Editor window.

1. Click either the Start or End button to move to start or end of the track line.
2. Click Skip Forward or Reverse.

Menu	Tools > Side Scan Editor > Playback Controls>Skip Forward/Skip Reverse
Tool	

View contacts while scrolling

If you want your contacts to remain visible in the waterfall display while scrolling or using the Playback functions

1. Select Side Scan Editor from the Tools > Options > Display tab.
2. Select the *Display contacts during playback* check box, and click Apply.

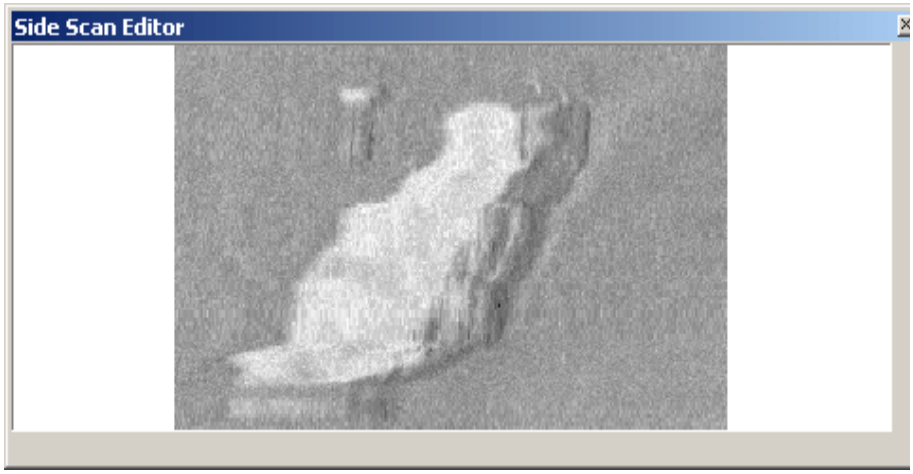
Zoom

You can magnify an area in the Side Scan window with the Side Scan Editor Zoom tool.

Menu	Tools > Side Scan Editor > Zoom
Tool	

1. Select the Side Scan Zoom tool.
2. Press and hold the mouse button and drag the cursor over the area you want to enlarge.

A new window displaying the enlarged area will open.



You can display the image so that the entire zoomed area fits the zoom window or you can scale the image according to a selected percentage.

3. Right-click the mouse in the zoom display window and choose any of the following options from the pop-up menu:
   - Select the *Fit to Window* option so that the entire area of the zoomed image is displayed in the window.
   - Select the *Aspect Ratio* to keep the same height and width ratio for the zoomed area. When this option is cleared, the image is resized so that it fills the entire zoom window.
   - Select the *Scale* option to resize the zoomed area to a scale. The scale is selected from the drop-down list. When a scale is selected, the image is displayed according to the selected percentage.
   - Select *Close* to close the window.
4. You can use the Measure Distance and Measure Shadow tools in the Zoom window (see "MEASURE DISTANCE" ON PAGE 509 and "MEASURE SHADOW" ON PAGE 510). When you release the mouse button, the results are shown in the Contacts tab in the Control Window.
5. Deactivate the Zoom tool when finished.

Selection

Data can be selected in the Side Scan Editor using the Side Scan selection tool. The selection tool can be used to select a range or a single ping within that range. Selected data can be viewed, edited, or corrected using a variety of tools.

To range-select data:

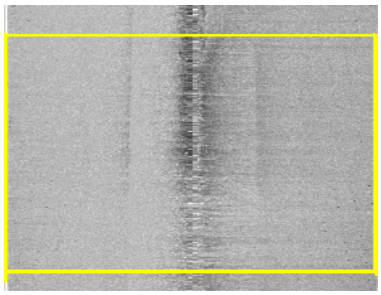
Menu	Tools > Side Scan Editor > Select
Tool	

1. Choose the Select Side Scan button on the Side Scan Editor toolbar.

The cursor will become a cross-hair when positioned in the Side Scan Editor window.

2. Press the mouse button and drag the cursor across the area of the Editor that you want to select.

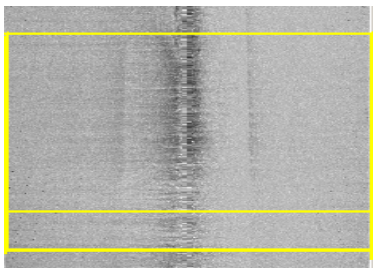
When you release the mouse button, the selected range is shown bounded with a rectangular yellow box.



The range remains selected until you click in the Editor window again.

To select a single ping:

3. Select a range of side scan (see above).
4. While the range is selected, press the <Up Arrow> key, which produces a third horizontal line representing the selected ping.

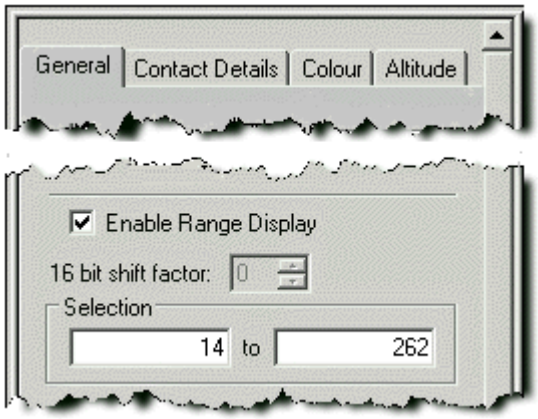


5. Continue pressing the <Up Arrow> key to move the selection up, or press the <Down Arrow> key to move the selection down.

Select by ping number

You can also use the Selection fields on the General tab of the Side Scan Editor Control window to select a range in the Editor window.

When you make a selection in the Editor window the start and end ping numbers of that selection are displayed in the General tab. To refine your selection, enter the specific ping numbers fields to specific start and end pings.



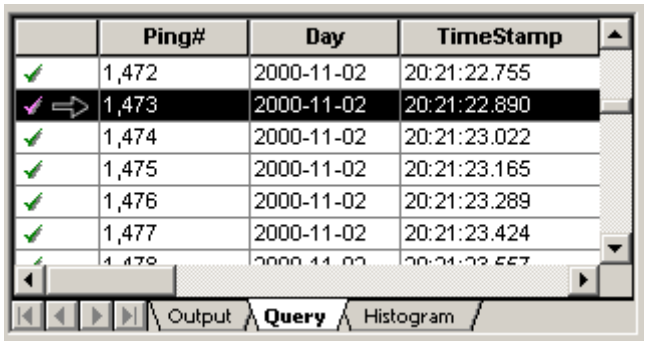
Superselection

Superselection is selecting data within an already selected range. You can superselect pings from the Worksheet window and display the superselected data in the Side Scan Editor.

1. Select a range of side scan data.
2. Select the Query command.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

A table listing the pings in the selection is displayed in the Query tab of the Worksheet window.

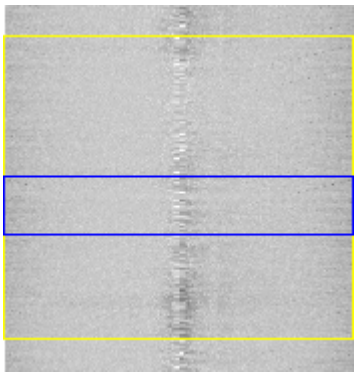


3. To superselect a single ping, select the row in the Query tab.

The superselected ping will be highlighted within the selection in the Side Scan Editor.

3. To superselect a range of pings, click in the first column on the first row of the range and drag the mouse through to the last row of the range you want to select.

A rectangle is displayed containing the superselected range of pings within the selection.



Colour Mapping

Colour maps

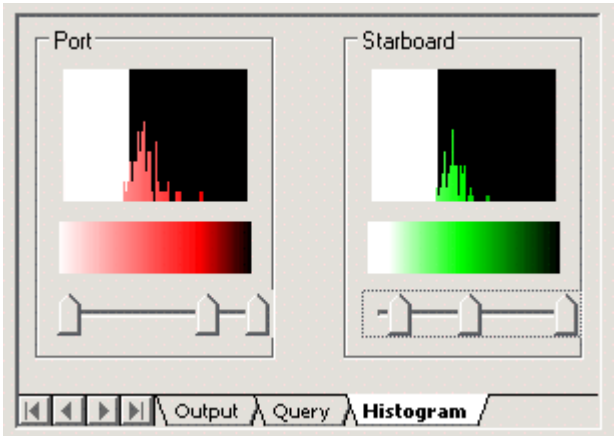
Colour maps provide a clear visual representation of sonar intensity levels along the track line. Colour mapping is applied to the histograms displayed in the Worksheet window, as well as the port and starboard sections of ping in the Side Scan Editor window.

You can select different colour maps for port and starboard sections, or select a single colour map for both sections. All colour maps are stored with a .cma extension in the directory …\Hips\System.

To change the colour mapping from the default setting of GreyScale:

1. Open the Control window and select the Colour tab from the Side Scan Editor tab.
2. Select either *Port*, *Starboard* or *Both* to apply new colour to.
3. Select a colour map from the drop-down menu (the default colour map is GreyScale.cma).

When the colour map is chosen, both the histogram and the ping window are refreshed, so that the port and starboard sections (or both) display the selected colour mapping.



In the Histogram, the horizontal axis represents the intensity range of the pixels in the sonar file and the vertical axis represents the distribution of pixels in the editor.

Use the slider bar below the histogram in the Worksheet window to adjust colours. Each colour in the histogram has a slider control associated with it.

1. Move the slider to adjust the pixel value of that colour in the histogram and in the ping.

The value shown in the Control window adjusts as you move the sliders.

Colour display can be adjusted also using the other options:

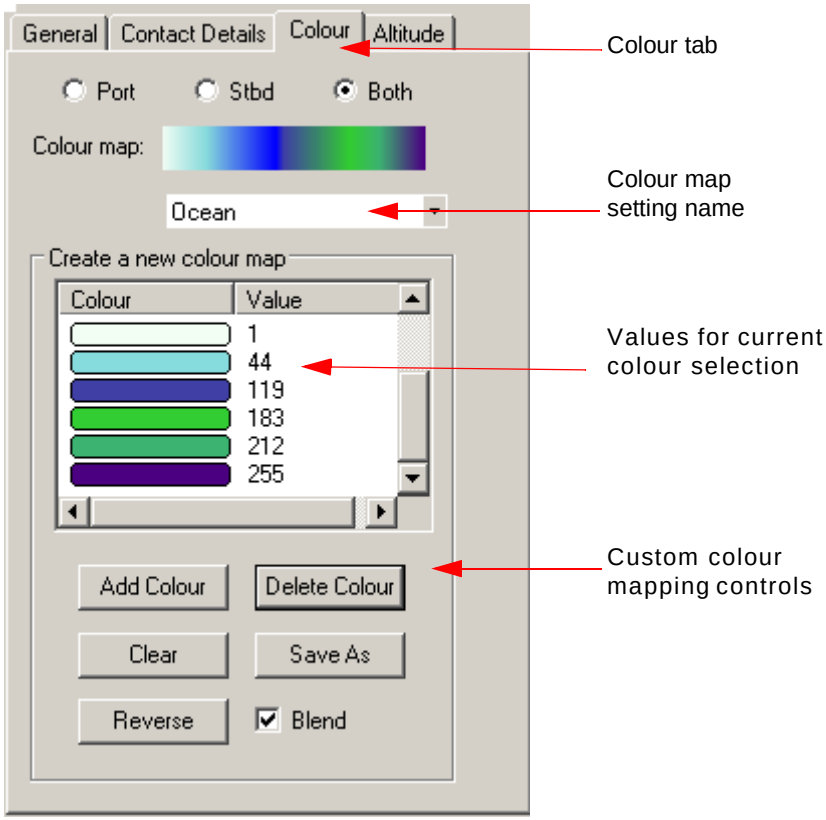
- **Blend** will blend the colour borders
- **Delete Colour** will remove it from the list, and from the areas that it colours
- **Clear** will remove all colours. To view ping clearly, restore GreyScale colour map.

Custom colour maps

You can create custom colour maps using either the Colour tab or the sliders below the histogram in the Worksheet window.

1. Make sure both the Worksheet and Control windows are open.
2. Click **Add Colour**, or click on a slider.

The Colour dialog box is displayed.



The *Value* field shows the pixel value of the displayed colour.

The pixel values range from 1 to 255 with the first colour always assigned to pixel value 1. Pixel value 0 is always assumed to be transparent.

3. Enter a pixel value for the colour, as needed.
4. Click on the button to open a colour palette

5. Select a colour from the colour palette or click **Create Custom Colour** to create a custom colour from the standard Windows colour picker.
6. Click **OK**.

The new colour and its assigned pixel value is added to the listing in the Control window, and applied to the histogram in the Worksheet window. The colour of the ping in the Side Scan Editor also changes to display the new colour in the appropriate areas.

7. Repeat Steps 2 to 6 to insert more colours, as needed.

As you select more colours, the colour values and the depths are displayed in the Control window.

Reverse

Sonar recorders have traditionally used dark colours to represent high signal returns. You may want to use the Reverse option to change the order in the colour map to reflect this practice.

To reverse the colour map order, click **Reverse**.

8. Click **Save As** to retain the settings as a colour map.

The Save As dialog box is displayed.

9. Type a file name for the colour map.

Measure Distance

Measure distance

This tool enables you to measure distance between two points with the mouse. This feature is used in processed mode, in either the Side Scan Editor window or in the Side Scan Zoom window. Distance units are determined by the settings in the Display tab of the Options dialog box (see "UNITS" ON PAGE 95).

Menu	Tools > Side Scan Editor > Measure Distance
Tool	

1. Make sure the side scan display is in processed mode.
1. From the Side Scan Editor toolbar, select the Measure Distance tool.
2. Press and hold the mouse button, and drag the cursor across the Side Scan Editor. To measure at right angles from a point, press the <Shift> key while using the measure tool.

As you drag the cursor across the data, the distance is displayed at the cursor.

Measure Shadow

Measure Shadow

This tool enables you to determine the height of an object by measuring its shadow. This feature is only available in processed mode, and can be measured in either the Side Scan Editor or in the Side Scan Zoom window.

1. Make sure the side scan display is in processed mode.
2. Select the Measure Shadow tool.
3. Press and hold the mouse button, and drag the cursor *horizontally* across the Side Scan Editor.

Menu	Tools > Side Scan Editor > Measure Shadow
Tool	

The height measurement is shown in the side scan window.

Edit Side Scan Data

The Reject/Accept/Query tools in the Side Scan Editor function as they do in other HIPS and SIPS editors, except that you select a ping rather than individual soundings or beams.

You can change the status of an entire ping or change only the starboard or port sides.

1. From the Side Scan toolbar, choose the Side Scan Selection tool and select the range of pings (see "SELECTION" ON PAGE 504).
2. Select any of the commands from the following table

Menu	Button	Description
Status Flag > Apply Edit to > Both		Edit both the port and starboard sides of the ping. (This option is chosen by default).
Status Flag > Apply Edit to > Port		Edit the port side of the ping.
Status Flag > Apply Edit to > Starboard		Edit the starboard side of the ping.

Reject

When a ping is rejected in the Side Scan Editor, it is given a status flag that excludes it from further processing in SIPS. However, the data is not deleted and can be changed to Accepted status again.

Menu	Edit > Status Flag > Reject
Tool	
Pop-up	Reject
Key	<R>

1. From the Side Scan toolbar, choose the Side Scan Selection tool and select the range of pings to be rejected (see "SELECTION" ON PAGE 504).
2. Select the Reject tool.

Accept

To make the rejected data available for further processing in SIPS, it must be restored using the Accept tool.

Menu	Edit > Status Flag > Accept
Tool	
Pop-up	Accept
Key	<A>

1. From the Side Scan toolbar, choose the Side Scan Selection tool and select the rejected pings (see "SELECTION" ON PAGE 504).
2. Select the Accept tool.

The previously rejected data is now flagged as accepted and is visible in the Side Scan Editor.

Query

Use the Query function to view information about selected pings.

Menu	Edit > Query
Tool	
Pop-up	Query
Key	<Q>

1. From the Side Scan toolbar, choose the Side Scan Selection tool and select a range of pings to be queried (see "SELECTION" ON PAGE 504).
2. Select the Query tool.

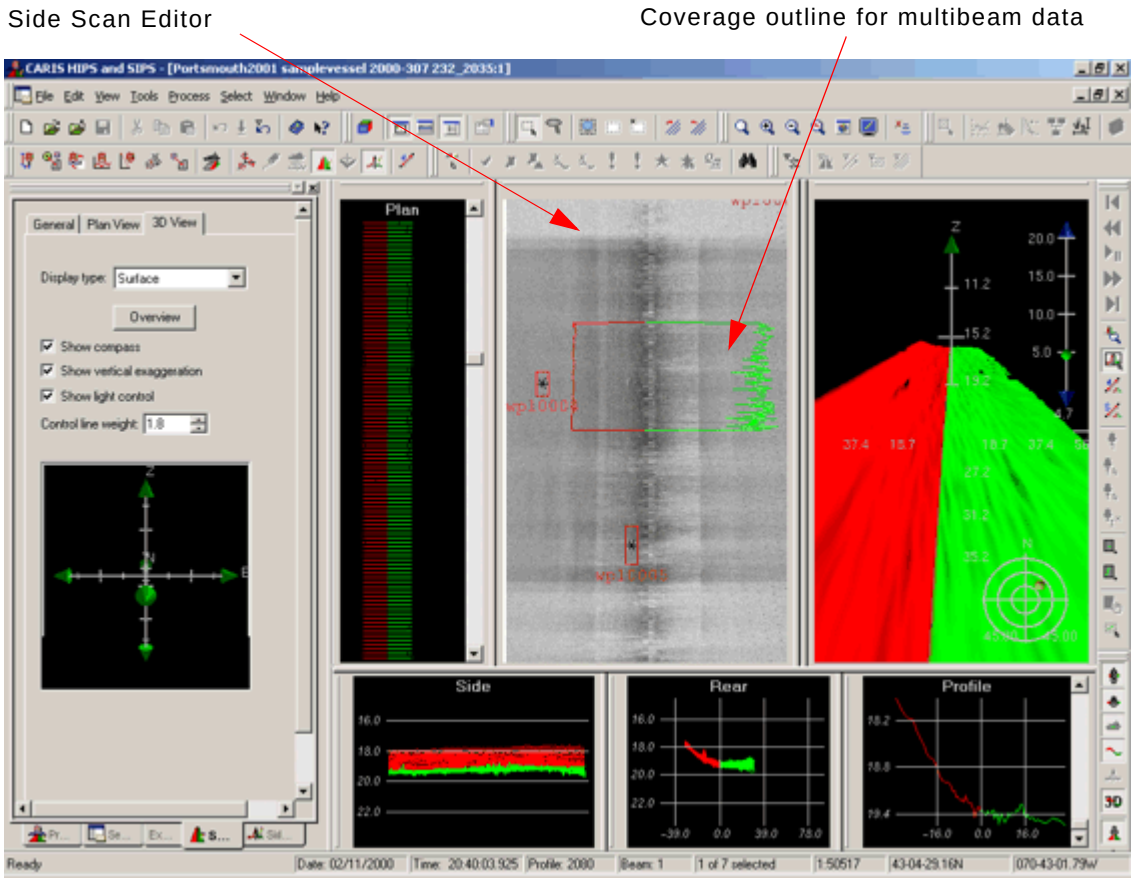
Information about the selected data is shown in the Worksheet window. If the window is not open at the time you query the data, it will open automatically.

Auto Cursor

You can combine the select and reject/accept/query functions with the Auto Cursor. See .

Side Scan Editor and Swath Editor

The Swath Editor can also be open at the same time as the Side Scan Editor.



When the Side Scan is open in this mode, the playback functions used to scroll the track line, are unavailable. You can, however, move up and down the track line by scrolling the Plan View in the Swath Editor (see "EXAMINE SWATH DATA" ON PAGE 307).

All other Side Scan functions and tools are available.

When both the Side Scan and Swath Editors are open, a coverage outline of the multibeam data is displayed in the Side Scan Editor. This feature is useful when the across-track coverage of the multibeam is less than that of the side scan.

Swath Time Offset

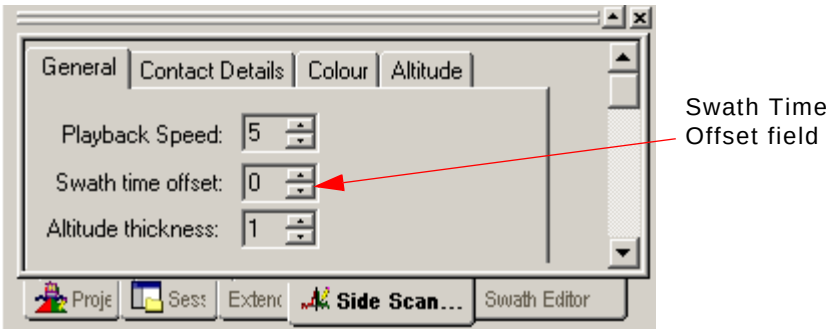
If the multibeam data was recorded from a vessel and the side scan from a tow fish, then the coverage outline will not be in the correct along-track location.

The *Swath Time Offset* adjusts the along-track locations of the swath outline in the Side Scan Editor by applying a time correction value. The time correction value can be either positive

or negative. Negative values are used for track lines where the side scan sonar is positioned before the multibeam echo sounder.

To adjust the swath time offset:

1. Make sure the Side Scan Editor is open. (The Side Scan Editor tab in the Control window will be visible.)
2. Click the <Up> or <Down> arrows for the Swath Time Offset field on the Side Scan editor tab until the correct delay (in seconds) between the vessel and towfish is displayed.



Signal Display Window

The Side Scan Editor includes a Signal Display Window for viewing sonar intensity levels. This window can be used as a comparative tool when performing corrections. The Signal Display Window can be used in both the raw and processed modes.

The Signal Display Window can display a mean of all sonar intensities in a selection or the sonar intensity values for a single ping. The display can be switched back and forth between mean and single profile modes by use of the buttons on the toolbar.

When the Signal Display Window is reopened, it will remember its previous setting and display either the mean intensity values for the entire selection or the intensity values of a single profile.

Open Signal Display Window

1. Select a range in the Side Scan window using the Side Scan Selection tool (see "SELECTION" ON PAGE 504).
2. Select the Signal Display Window tool.

Menu	Tools > Side Scan Editor > Signal Display Window > Open/Close
Tool	

The Side Scan Signal Display Window is displayed.

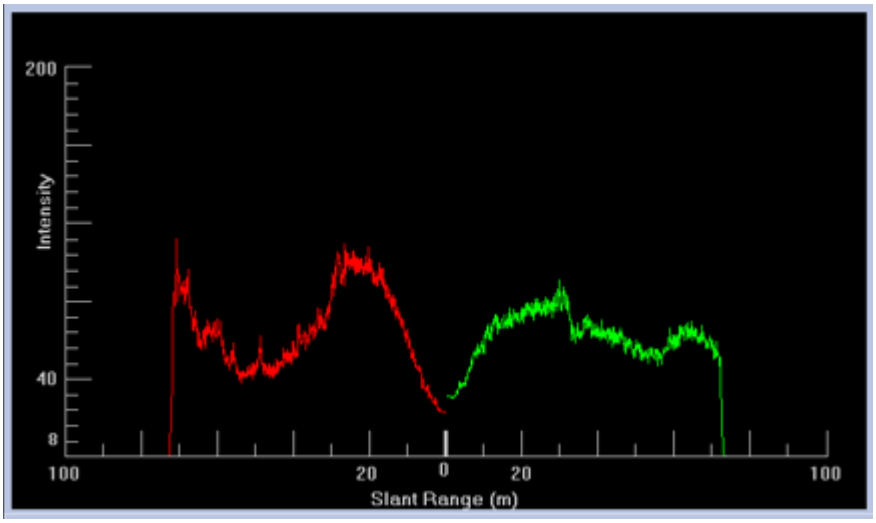
The vertical axis on the graph represents intensity level of the signal(s) being examined. The horizontal axis represents either time or slant-range distance depending on the display mode of the Side Scan Editor.

View Selection Mean

The mean of the intensity values in the selected range can be displayed.

3. Select the View Selection Mean tool.

Menu	Tools > Side Scan Editor > Signal Display Window > View Selection Mean
Tool	



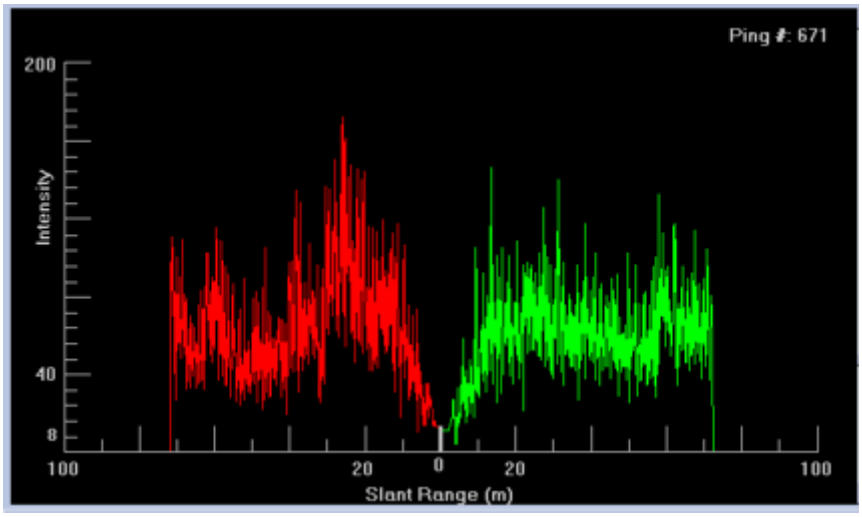
View Single Ping

You can also view signal intensity for a single ping.

Menu	Tools > Side Scan Editor > Signal Display Window > View Single Ping
Tool	

4. Select the View Single Ping tool.

The Signal Display Window is refreshed to show the sonar signal intensities for the single ping selected in the Side Scan window.

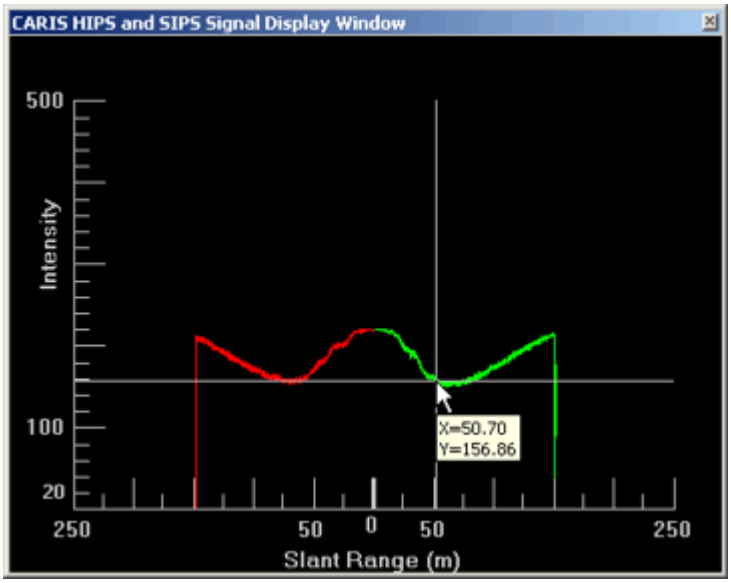


Pressing the Up Arrow key while an area of the side scan data is range selected will select a single ping

5. Use the Up and Down arrow key to view the sonar intensities for other single pings.

As you scroll through the selected range, the Signal Display Window will refresh to show the intensity values for each ping. The ping number is displayed in the top-right corner of the Signal Display Window in this mode. The location of the ping is also indicated by a new line within the selection. The single ping selection always defaults to the first ping in the selection.

You can view the position and specific intensity values of points by clicking on any point in the graph in the Signal Display Window. The intensity and time/distance values for that point are shown in a tool tip.

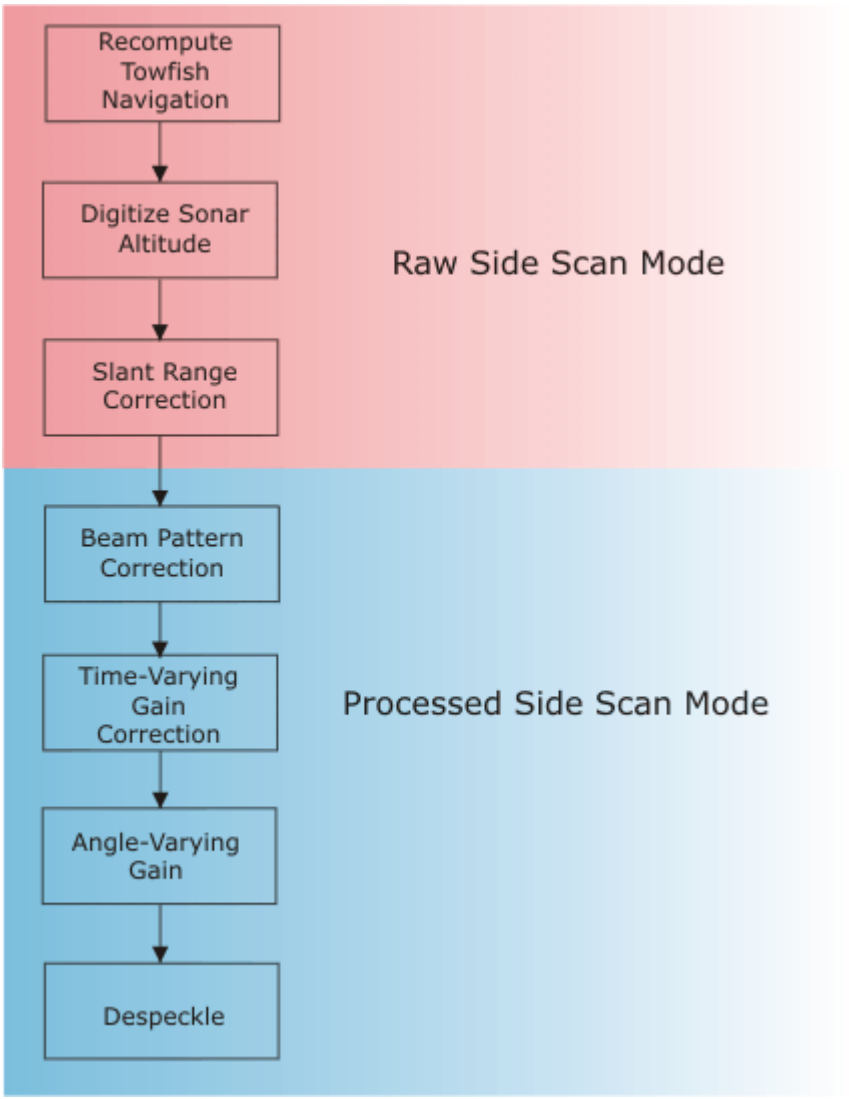

CARIS HIPS and SIPS Signal Display Window
500
Intensity
100
20
X=50.70
Y=156.86
250
50
0
50
250
Slant Range (m)

Processing

Side scan data can be processed using a number of carefully designed process tools. These tools can be divided into two groups based on the display mode in which they can be used: raw or processed.

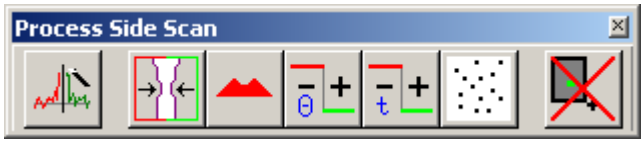
The following diagram displays the suggested order to apply the corrections and the side scan mode in which they can be used.

Processing can be accomplished using the Side Scan Editor as a selection and visualization tool, however, if the Side Scan Editor is closed you can also apply corrections to one or more track lines in the main Display window.



Processing can be accomplished using the Side Scan Editor as a selection and visualization tool, however you can also apply correction to one or more track lines in the main Display window.

Correction tools that create or modify processed side scan data may be accessed from the Side Scan Processes toolbar.



The following processing operations are available in this application:

"Recompute Towfish Navigation" on page 519

"Digitizing Sonar Altitude" on page 521

"Slant Range Correction" on page 525

"Apply Slant Range Correction" on page 526

"Beam Pattern Correction" on page 529

"Angle-Varying Gain" on page 534

"Time-Varying Gain" on page 535

"Despeckle" on page 538

"Editing Contacts" on page 542

Recompute Towfish Navigation

Navigation data for towed sensors can come from the following sources:

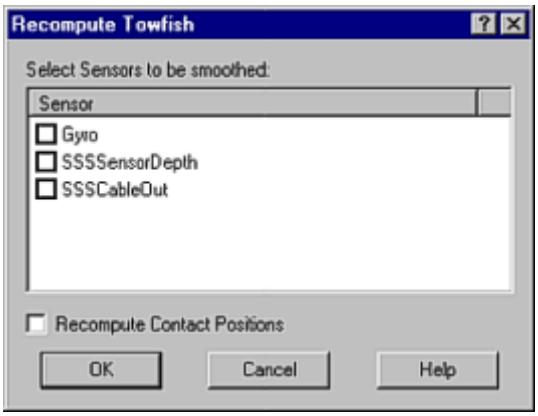
- If computed during data logging and stored in the raw side scan format, it can be converted directly into the SIPS towfish navigation data structure. The Recompute Towfish Navigation process is therefore not required.
- It can be computed by SIPS if specific sensor data is available. SIPS requires ship navigation data with cable out and towfish depth to compute towfish navigation, or, ship navigation with horizontal layback.

If the towfish navigation data will be computed by SIPS then:

1. Select one or more towfish track lines.
2. Select the Recompute Towfish Navigation command.

Menu	Process > Recompute Towfish Navigation

The Recompute Towfish Navigation dialog box is displayed.



Towfish sensor data can be smoothed in the Attitude Editor (see "SMOOTH DATA" ON PAGE 278) and this data can be applied when the towfish navigation is computed. Select a check box to apply the smoothed data.

3. Select the *Recompute Contact Positions* check box to update the contact positions based on the new navigation data.
4. Click **OK**.

If contacts are contained in the track lines where towfish navigation has been updated, you must update the contact positions to match the new navigation (see "RECOMPUTE CONTACT POSITION" ON PAGE 544).

To compute towfish navigation from the ship's navigation requires a horizontal distance and direction from the ship's towpoint location to the towfish. The towpoint is defined in the vessel file.

- **Horizontal distance**. Recorded horizontal layback data or actual cable length both share the same data structure in SIPS called SSS Cable Out. SIPS can distinguish between the two by the towfish sensor depth data. If the sensor depth data is zero, the horizontal layback is used. If the sensor depth data is non-zero it will be combined with the cable out data using trigonometry to produce horizontal distances. All distances to the towfish are applied from the towpoint position on the ship as defined in the vessel configuration file.
- **Direction**. The direction from the ship's towpoint location to the towfish is derived using a "follow-the-dog" method. The average of the first 30 seconds of the ship's "course-made-good" is used for the initial direction. Thereafter, each successive position computed for the towfish occurs along the direction from the next ship's position to the previous towfish position.

If horizontal layback or tow cable length data is not available, or if the Recompute Towfish Navigation step is not executed, then

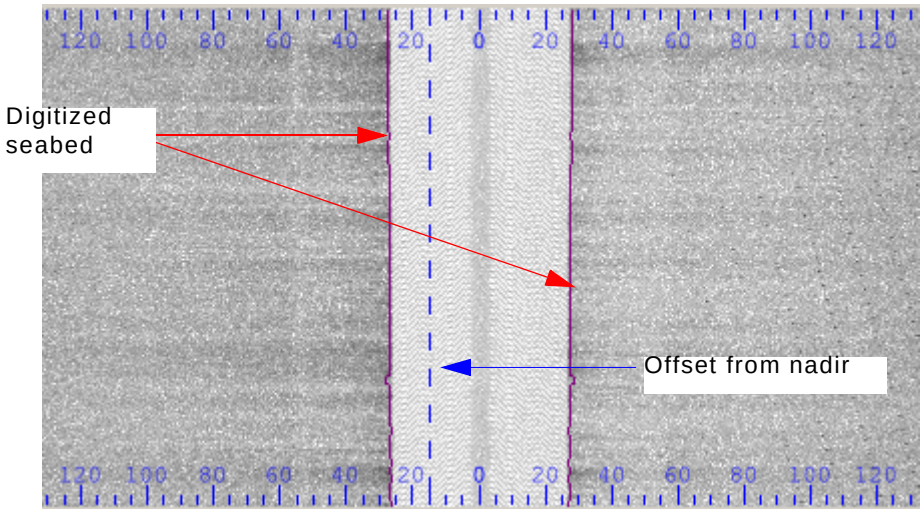
the recorded towfish navigation data converted into SIPS is used for generating the positions of contacts and for mosaics.

If towfish navigation is not available at all then the position of the side scan imagery data is assumed to be the same as the ship navigation data.

Digitizing Sonar Altitude

Before side scan data can be processed, the altitude of the sonar must be known. With some sonars, altitude is logged in the raw data files and is converted by SIPS. However, in other cases, altitude is not logged or is logged and needs to be edited. The Side Scan Editor provides automated and manual tools to digitize the bottom directly from the side scan trace.

A raw side scan sonar display with a digitized seabed is shown below.



Digitization and colour maps

Sometimes there is no clear distinction between the water column and the seabed. You can compensate for this by adjusting the display of intensity values in the editor so that the seabed and water column are clearly outlined. Also see “COLOUR MAPPING” ON PAGE 507.

1. Make sure that only two colours are displayed in the editor (for example, GreyScale.cma).
2. Use the sliders in the colour histogram to adjust the display in the editor.
3. Once the there is clear distinction between the water column and seabed use the digitizing tools to trace the first bottom return.

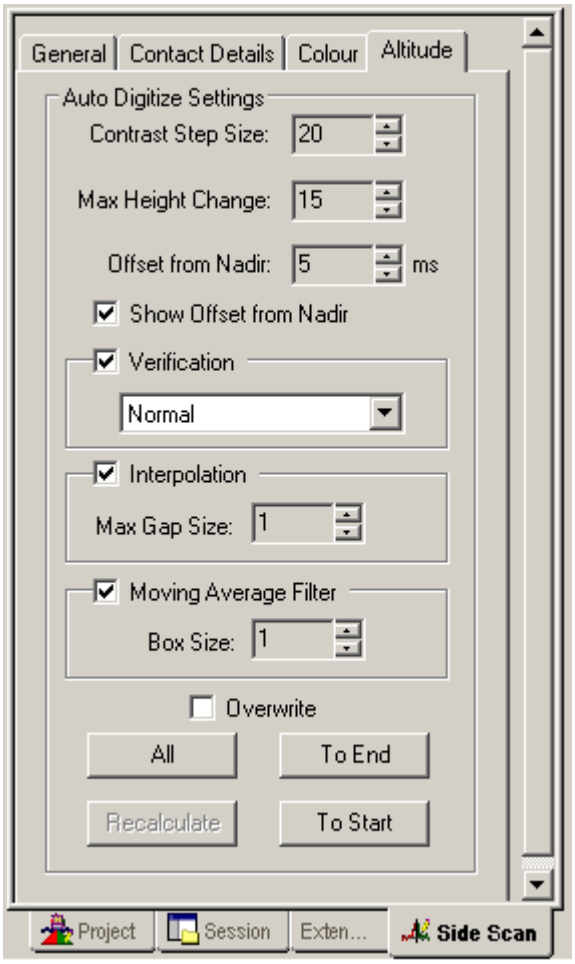
4. To remove the digitized lines, select the Clear command.

Menu	Tools > Side Scan Editor > Altitude > Clear
Tool	

Automatic digitizing

The automatic digitizing method traces the first bottom return using a sophisticated multi-pass algorithm. Automatic digitizing is most effective when there is a clear distinction between the water column and the bottom. This method can be applied to a range of pings or to the entire line.

Settings for automatically digitizing the side scan return are in the Altitude tab of the Control window.



The Altitude tab options are described below in the order that they appear in the tab. While the *Contrast Step Size* and *Max Height Changes* are compulsory for digitizing, the other options can be applied depending on the specific conditions of your data.

Contrast Step Size

The *Contrast Step Size* and *Max Height Change* options must be set to correctly auto digitize the side scan.

The *Contrast Step Size* is the minimum difference in contrast expected between the intensity values of the water column and the seafloor. The Signal Display Window may be helpful in determining a good step size.

Max Height Change

The *Max Height Change* is the maximum allowable ping-to-ping change in detected bottom position (in pixels). It is a restriction on the variation anticipated in detected bottom position. It could be visualized as error bars guiding the algorithm along the bottom.

1. Use the <Up> or <Down> arrow buttons to select the percentage for the *Contrast Step Size* and *Max Height Change* values.

Offset from Nadir

Strong solid returns in the water column, such surface reflections, can be compensated for by setting an offset from nadir in raw data mode.

2. Set the value in milliseconds for an *Offset from Nadir* by clicking the up or down arrow buttons.
3. Select *Show Offset from Nadir* to display the offset visually as a dashed line.
   - The line colour is set to the *Measure / Offset tools* colour in Tools > Options > Side Scan Editor.
   - The thickness of the line is set in the *Line thickness* field on the General tab in the Side Scan Editor Control tab.

Verification

The verification option tests the detected point to see if its intensity is consistent with nearby values further across-track that should also represent seafloor.

Its purpose is to detect and ignore anomalous points in the water column (for example, noise from equipment, reflections off the surface, marine life), by checking to see if there is water rather than sea floor on the other side of a detected point.

4. Select the Verification check box.
5. Select a desired strength-level for the verification from the drop-down list. Choices range from “Very Weak” to “Very Strong”. The default is “Normal”.

Interpolation

The *Interpolation* option fills along-track gaps in the detected bottoms.

6. Select the *Interpolation* option.
7. Select the size of the gap to be filled by clicking the up or down buttons beside the *Max Gap Size* field.

Moving Average Filter

The *Filter* option smooths the digitized line.

8. Click the Moving Average Filter check box.
9. Click the <Up> or <Down> arrow buttons to select the number of pings used for averaging in the Filter Width field.

Menu	Tools > Side Scan Editor > Altitude > Auto Digitize
Tool	

You can use the Auto Digitize tool, or use the buttons at the bottom of the Altitude tab to complete digitizing. If you are using the buttons, continue with Step 8; otherwise, go to Step 11.

10. Choose any of the following digitization options:
    - Click **To Start** to extend the digitized region to the back of the entire line.
    - Click **To End** to extend the digitized region to the end of the line.
    - Click **All** to digitize the entire selection or the entire line.
11. If you change a value in the tab, click **Recalculate** to update the Digitization.
12. Select *Overwrite* to enable re-digitization of selected areas without clearing. With Overwrite disabled, only zero-altitude/cleared fish heights will be available for digitization.
13. Click anywhere on the seabed to select the start and end pings for digitization and also to indicate the side (port or starboard) on which to perform the calculations.
14. [Optional] You can select the Clear tool to remove the digitization.

Menu	Tools > Side Scan Editor > Altitude > Clear
Tool	

Manual digitizing

Add altitude lines by selecting areas along the first bottom.

Menu	Tools > Side Scan Editor > Altitude > Digitize
Tool	

1. Select the Manual Digitize tool.
2. Press and hold the mouse button and drag the cursor along the bottom contact.

> You can also use the space bar to move along the track line while in digitizing mode.

As you drag the cursor, a digitizing line is drawn in the window. When you digitize one side, the other side is automatically mirrored.

3. Alternatively, create a point-to-point line by clicking carefully along the first return.
4. Cancel the Manual Digitize tool when finished.

At any time during the digitizing process, you can select the Clear tool to remove the digitization.

Menu	Tools > Side Scan Editor > Altitude > Clear
Tool	

Interpolate Selection

Interpolation digitizes a straight line between the fish height values found at the start and end of the selection. Any altitude data within the selection is overwritten during this operation.

1. Use the Side Scan Selection tool to select an area of raw mode data that has been digitized. (See "SELECTION" ON PAGE 504)
2. Select the Interpolate Selection command.

Menu	Tools > Side Scan Editor > Altitude > Interpolate Selection

The pixels between the digitizing points are interpolated and the digitizing lines are smoothed.

Slant Range Correction

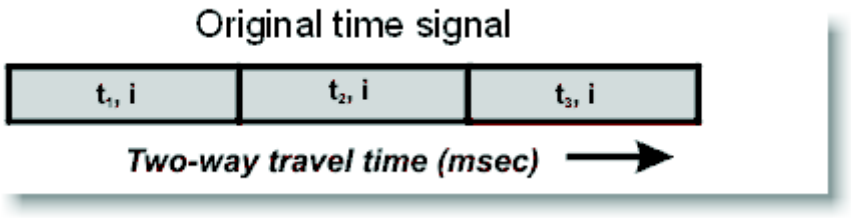
Side scan sonar is initially logged as a series of time-indexed intensity values for each ping. Slant range correction re-samples this raw side scan data to produce profiles of image pixels where the across-track scale is related to distance from nadir instead of time.

Across-track positioning using height sources

The standard slant range correction operation applies a sound velocity estimate to the two-way travel time for each intensity value. This produces an estimated ray length that when combined with the known sonar height (from the digitized or logged bottom) is used to produce the across-track distance to the pixel using simple trigonometry. This method presupposes that the bottom is flat across the ping and can result in the across-track misplacement of objects over varied or steeply sloping terrain.

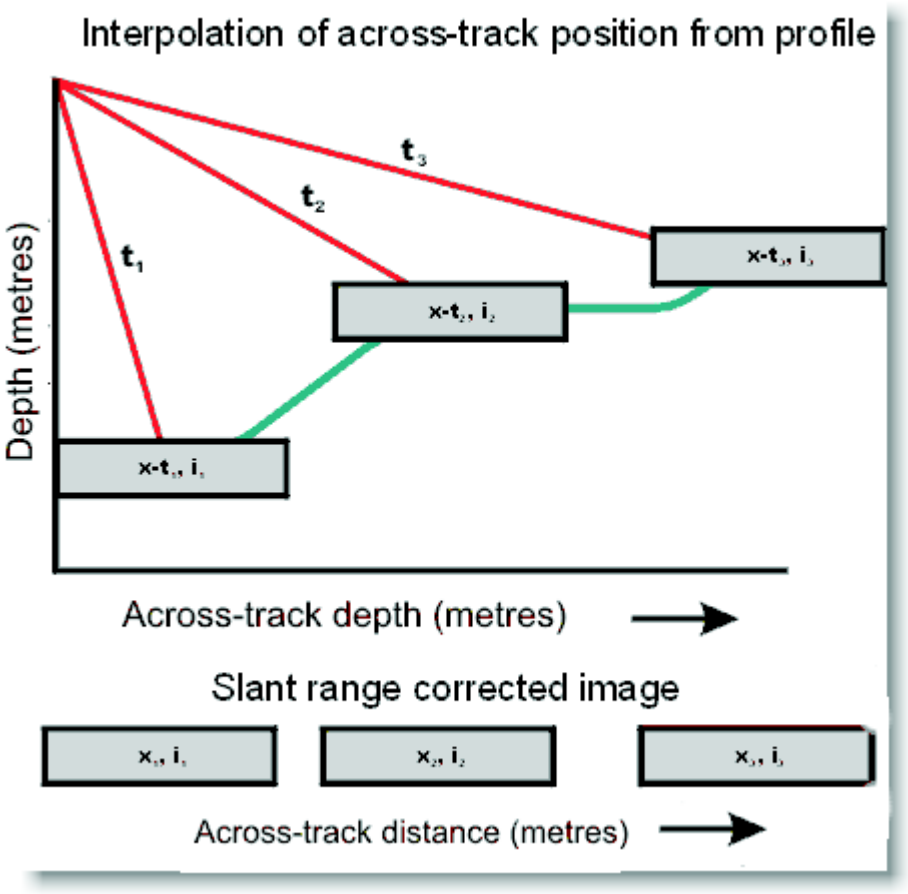
SIPS can use a height source (such as a BASE Surface, grid, or tile) in the slant range correction operation to supply depth values to improve the across-track positioning of pixels.

In the following figure, raw side scan data is represented as a series of time (t) and pixel intensity values (i).



When a height source is used during slant range correction, a depth profile is first derived from the source along the path of the

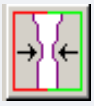
side scan ping. Then, the two-way travel time to each pixel is converted to a distance from nadir using the estimated velocity of sound in water. The distance is then used to calculate the position of the image pixel along the ping drawn from the height source.



The resulting output is an intensity value indexed to across-track distance (x) rather than time.

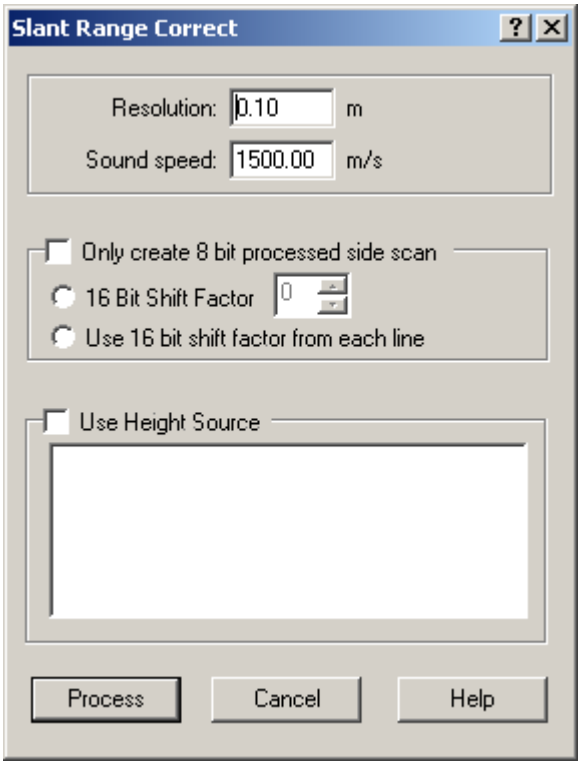
Apply Slant Range Correction

1. Select the track line(s) in the Display window.

Menu	Process > Slant Range Correct
Tool	

2. Select the Slant Range Correct tool.

The Slant Range Correction dialog box is displayed.



3. Type the resolution (in metres or feet) for the slant range corrected file. The resolution value defaults to the minimum value appropriate to the sonar system. If this is not available then it defaults to 0.10 metres or 0.33 feet. A finer resolution generates a larger file.
4. Enter the average speed of sound.
5. Select the *Only Create 8-Bit Processed Side Scan* check box if you want to create a slant range file using only eight bits of storage per pixel. This option is important if the data that you are slant range correcting was originally 16-bit and all 16 bits were maintained at conversion. Otherwise it has no effect.

You can use the existing 16-bit factor that is stored with the line or adjust the bit factor and then slant-range correct the data.

6. To apply a new bit factor to the data, select the *16 Bit Shift Factor* option and select the bit factor by clicking the arrow buttons beside the bit factor size box.
7. To apply an existing bit factor, select the *Use 16-bit Factor From Each Line* option.
8. Select the *Use Height Source* check box if you want to apply a height source, and select the height source.
9. Click **Process**. The slant range corrected data is displayed in the Side Scan Editor.

Revert/Reapply

The most recent correction made in *processed* mode can be undone or redone using the Revert and Reapply tools.

1. Select the tool to reverse the last correction.
2. Select the tool to reapply the last correction.

This facility to go back a step without losing your work, in effect enables you to "toggle" between the before and after of a process. For example, with the Signal Display Window open, use Revert/Reapply to see the effect of slant range correction.

Menu	Tools > Side Scan Editor > Revert
Tool	

Menu	Tools > Side Scan Editor > Reapply
Tool	

Beam Pattern Correction

Beam pattern correction uniformly removes along-track banding inherent to the sonar beam. This effect is produced by each transducer uniquely.

Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect. Tools are provided to create, view and process beam pattern files. The process uses the beam pattern file to uniformly correct the selected line(s), producing a more consistent appearance in mosaics.

Once the beam pattern file is created it can be applied to one or more lines in the Display window, or to one or more pings in the Side Scan Editor.

Create Beam Pattern Correction File

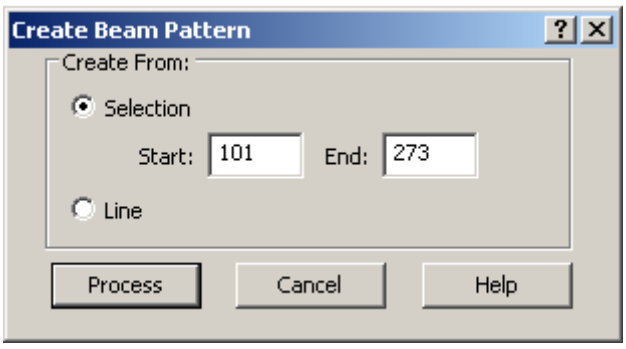
All the tools to create a beam pattern file are accessible on the Beam Pattern Toolbar, while the Side Scan Editor is open.

To create a beam pattern file you must select an area that is as flat, featureless and homogenous as possible. It is recommended that a "patch test" be done during a survey over an area known to have such properties. In order to create a more robust and accurate beam pattern correction file, it is also important to vary the fish height over the range expected to be used during the survey.

1. Using the Side Scan selection tool, select an area along the track line.
2. Select the tool.

Menu	Tools > Side Scan Editor > Beam Pattern Correction > Create
Tool	
Pop-up	Create Beam Pattern…

The Create Beam Pattern file dialog box is displayed.



The option is available to alter the range of pings to be included in the creation of the beam pattern. While this should be done with care, it is possible to create a beam pattern file from a selection larger than the range visible in the Side Scan Editor

display. You may also select the whole line for the creation of the beam pattern if you wish. It is recommended that only an area meeting the criteria set forth above be considered for inclusion in a beam pattern file.

3. Select **Process** to create the Beam Pattern file.

The Windows Save As dialog box is displayed.

4. Type a name for the Beam Pattern file and select **Save**.

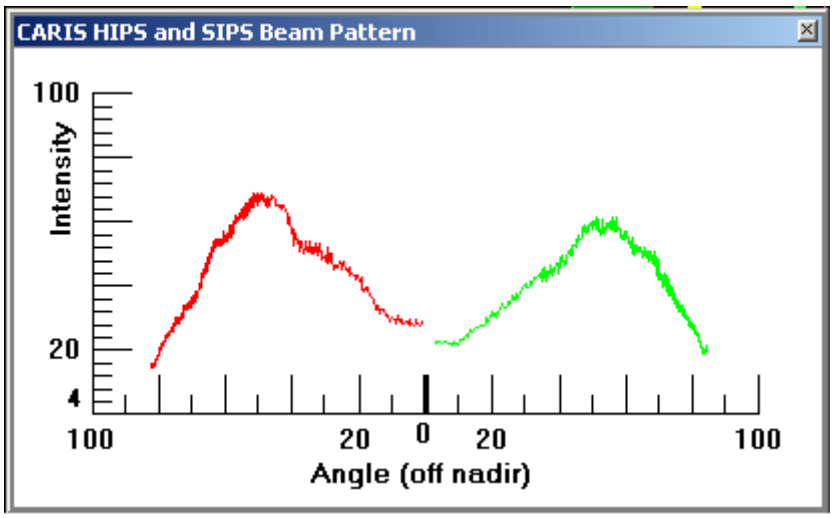
The **Save Beam Pattern**, **Save Beam Pattern As**, **View**, **Remove Spikes**, **Smooth,** and **Reload** tools are now active in the menu and on the toolbar.

View Beam Pattern

Menu	Tools > Side Scan Editor > Beam Pattern Correction > View
Tool	

5. Select the tool.

The Beam Pattern Graph window is displayed.This window shows the intensity levels of the side scan sonar over the selected area.



Remove Spikes

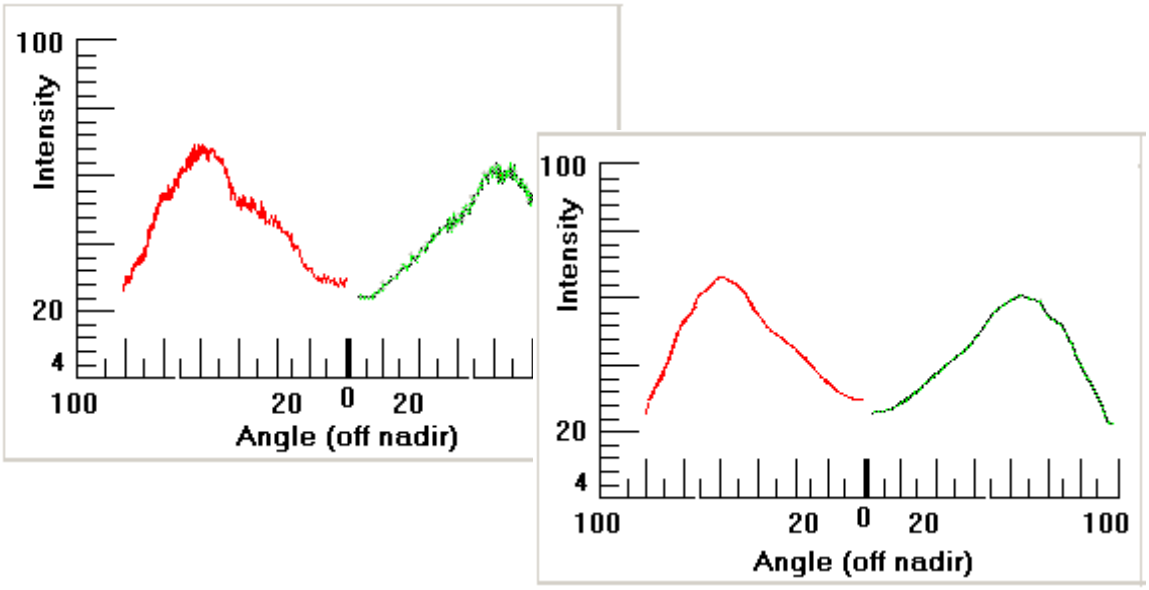
The Remove Spikes tool removes anomalies from the Beam Pattern Graph.

6. Select the Remove Spikes tool.

Menu	Tools > Side Scan Editor > Beam Pattern Correction > Remove Spikes
Tool	

The next step is to apply the Smooth Beam Pattern tool. The Smooth Beam Pattern tool uses a moving average filter to reduce noise in the beam pattern.

An example of the graph before and after using the Smooth tool is shown below. The left-hand window shows the unsmoothed graph and right-hand window shows the graph after smoothing.



7. Select the tool.

Menu	Tools > Side Scan Editor > Beam Pattern Correction > Smooth
Tool	

The Smooth Beam Pattern tool can be repeated until the preferred degree of smoothing is reached.

Reload

The **Reload** tool reloads saved changes from the active beam pattern file. Any changes to the beam pattern before the last save are lost.

For example, if you remove spikes and then Save, then apply Smooth Beam a few times, the Reload tool will restore the graph to the point just after you saved it.

Menu	Tools > Side Scan Editor > Beam Pattern Correction > Reload
Tool	

8. Select the Reload tool.

Save/Save As

When you have completed smoothing the data, you can save these settings using the Save Beam Pattern or Save Beam Pattern As… tools.

Menu	Tools > Side Scan Editor > Beam Pattern Correction > Save
Tool	

9. Select Save Beam Pattern to save the filtering and smoothing changes to the Beam Pattern File.
10. Select the Save Beam Pattern As command to save the file as a new file.

Apply Beam Pattern Correction

The Beam Pattern Correction file can be applied to a selection in the Side Scan Editor or to track line(s) in the Display window.

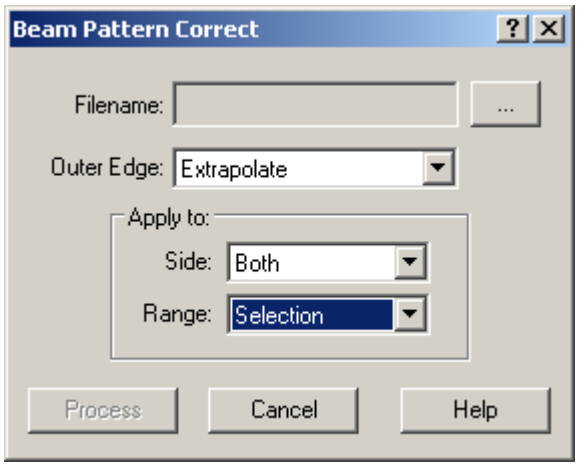
1. Make a selection in the Editor, or,

1. Select a track line or group of track lines in the Display window.

2. Select the command.

Menu	Process > Beam Pattern Correction
Tool	

The Beam Pattern Correction dialog box is displayed.



The Beam Pattern file will default to the last created, loaded, saved or executed one. You can use the Browse button to locate a different file for processing. The default location for Beam Pattern files is <Data Dir>/Sidescan.

The outer edges of some pings may have been recorded at angles beyond the extent of the beam pattern file being used. The Outer Edge option must be specified for handling these cases.

3. Select a method for handling sonar data collected outside the bound of the beam pattern file:"
    - Extrapolate: Extrapolate the beam pattern to correct angles beyond the outer edge of the generated beam pattern file.
    - Trim: Remove sonar data that is beyond the extent of the beam pattern file. Trimmed data will appear as 0 intensity white space in the Side Scan Editor.
    - Do Nothing: Does not correct sonar data beyond the extent of the beam pattern file. Uncorrected data will appear the same before and after Beam Pattern Correction in the Side Scan Editor.
4. Apply the Beam Pattern Correction to the both sides of the side scan or to either the starboard or port sides.
5. You can apply the Bean Pattern Correction to your entire survey by selecting all the track lines, or you can open the Side Scan Editor and select a portion of the data:

- *Ping*: the correction will apply to the single ping selected in the Side Scan Editor window.
- *Selection*: the correction will apply to the range of pings you selected in the Side Scan Editor window.
- *Line*: the correction will apply to the entire trackline you have selected.

6. Click **Process** to apply the correction.

Angle-Varying Gain

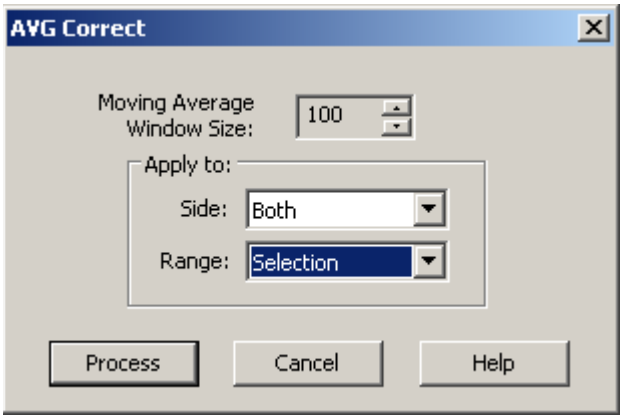
Angle-Varying Gain (AVG) correction removes the angular response of sediment from the imagery, normalizing the mean angular intensities, ping-by-ping, with a moving average filter.

AVG correction can be applied to a selection or a single ping in the Side Scan Editor or to one or more track lines in the Display window.

1. Make a selection in the Side Scan Editor. (See "SELECTION" ON PAGE 504 for selecting areas and pings).
2. Select the AVG Correct tool.

Menu	Process > AVG Correct
Tool	

The AVG Correct dialog box is displayed.



3. Select the number of adjacent pings to include in the moving average by clicking *Moving Average Window Size* up or down arrow buttons.

For example, an average based on a window size of 100 will include 50 pings before and after the ping being corrected.

4. Apply angle-varying gain to the port, starboard, or both sides of the side scan.
5. Apply the angle-varying gain correction to either a *Ping*, *Selection*, or *Line* by selecting the appropriate option.
6. Click **Process**.

Time-Varying Gain

Time-Varying Gain (TVG) Correction allows you to adjust the signal intensity by applying a non-uniform, time-dependent gain.

During acquisition, this is necessary because of the attenuation that occurs due to absorption and spreading as the sonar beam travels back to the receiver. Since these returns are received over a predictable and constant time period, a time-varying curve can used to increase gain to compensate for the decayed sonar return. In post-production, it maybe necessary to augment the TVS correction applied during acquisition.

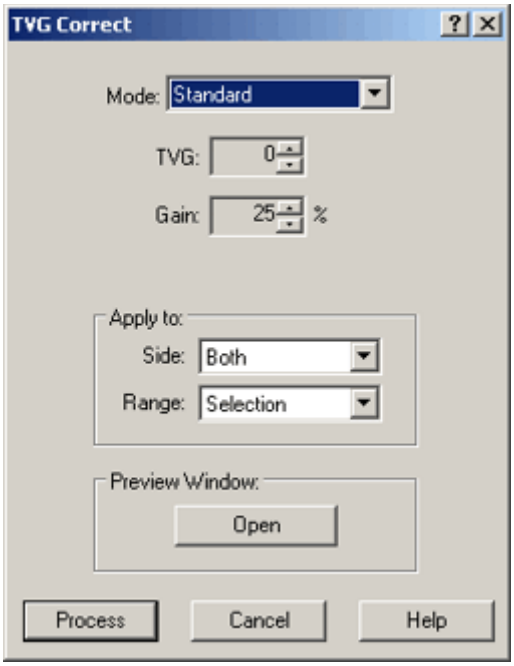
The TVG Correction command also allows you to apply a uniform gain correction without applying the any time-dependent gains by using only the Gain control.

TVG Correction can be applied to a single ping, selection, or the currently selected line in the Side Scan Editor; or to one or more track lines in the Display window.

1. Make a selection in the Side Scan Editor. (See "SELECTION" ON PAGE 504 for selecting areas and pings).
2. Select the TVG Correct command.

Menu	Process > TVG Correct
Tool	

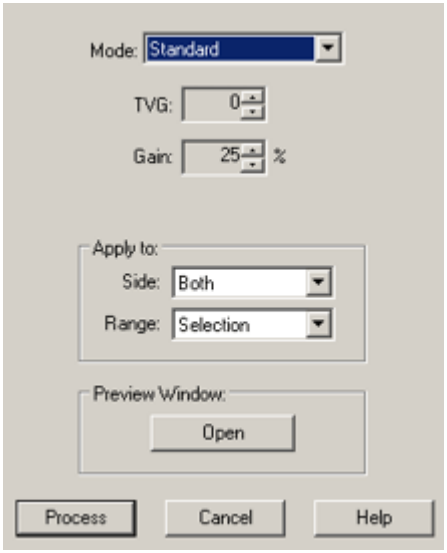
The Time-Varying Gain Correct dialog box is displayed.



There are two modes for TVG correction: Standard and Advanced.

3. Select either the Standard or Advanced mode from the *Mode* drop-down list.

The options available depend on the mode selected. The dialog box is refreshed to show the options.



The Preview window is available when a selection or superselection is made in the Side Scan Editor. The Preview window is not available when a track line is selected.

The Preview window is divided into three panes:

- Gain (shown as a percentage)
- Intensity values before TVG is applied.
- Intensity value after TVG is applied.

Changes are instantaneously shown in the Preview window.

4. Click **Open** to view the Preview window. To close the window, click **Close**.
5. The TVG correction can be made to port, starboard, or both sides by selecting the appropriate from the *Side* drop-down list.
6. The correction can be made to a single ping, a selection, or the entire line by selecting any of these options from the Range drop-down list. (The options depend on the selection made in the Side Scan Editor or in the Display window).

Standard

The Standard mode contains two options for correcting TVG.

7. Increase or decrease the percentage *Gain* by clicking the up or down arrow buttons.
8. Increase or decrease the *TVG* value by clicking the up or down arrow buttons. This changes the TVG curve.

Advanced

The Advanced mode contains more precise options for correcting TVG.

9. *Spherical Spreading* compensates for the loss of energy caused by signal displacement along the sea floor. Increase or reduce the signal intensity by clicking the Spherical Spreading up or down buttons.
10. The *Absorption* value compensates for attenuation due to energy loss in water. The scale factor is in dB/unit distances. Click the up or down arrow buttons to increase or reduce this value.
11. Click **Process** to apply the TVG correction.

Despeckle

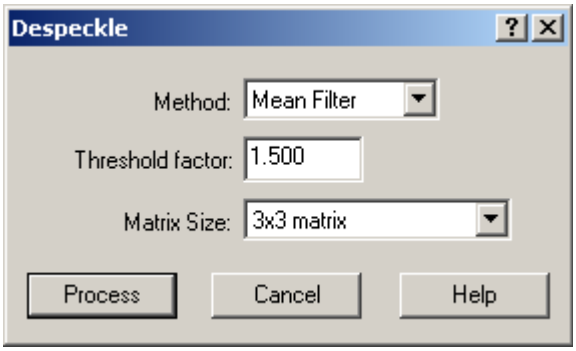
Despeckling uses a calculated value based on neighbouring intensity levels to replace the current pixel's intensity if it is outside the specified range. The value calculated from the neighbouring intensities can be derived by one of two methods mean or median.

The following steps are used for despeckling images:

1. Select a track line.
2. Select the Despeckle tool.

Menu	Process > Despeckle
Tool	

The Despeckle dialog box is displayed.



3. Select the Mean or Median Filter from the *Method* drop-down list.
4. Use the up and down arrows to select a *Threshold* for defining an acceptable range in which the pixel value will not be replaced.

For example, if the mean value is 50 and the threshold is +/- 20% then the acceptable range is 40 to 60. Thus a pixel value of 38 would be replaced by the mean value of 50 as it is outside the range, but a pixel value of 42 would not be replaced.

5. Select a pattern of neighbouring pixels from the *Matrix Size* drop down list be used in calculating the mean/median.

For example

6. Click **Process**

The line is filtered to remove noise from the sonar image.

Contacts

A Contact is the representation of an object or feature on the sea bottom (for example, a wreck, or a pipeline) that is visible in the sonar data. Contacts can be georeferenced, have positions calculated, and have attributes attached to them in the Side Scan Editor using the Add Point or Add Line Contacts tools.

In SIPS, you can pick individual contacts directly from the Side Scan Editor interface. A contact record is created for each contact, and includes sufficient information to be subsequently imported to a CARIS map or to a database application.

A contact file is created for each survey line and is stored in the line folder within the project directory structure. Each contact file is a text file that can be viewed and edited using a text editor. Each file consists of zero or more contact records.

For more information on contact file structure, see "Contact File Formats" on page 657.

Create Contacts

Point type contacts can be created for objects such as rocks, underwater posts or pile as well as wrecks. Contacts can also be created for continuous objects such as pipelines or underwater cables.

Contact information is entered on the General tab of the Control window and updated on the Contact Details tab.

1. Make sure the Control window is open and the General tab in the Side Scan Section is displayed.
2. Enter information in the appropriate fields on the General tab.
   - *Contact Type:* Select a point or line contact type from the appropriate drop-down list. Each contact type is assigned a CARIS feature code, and there is an identifying symbol for each type. The contact types shown in the lists and the corresponding CARIS feature codes are stored in …\Hips\system\ContactFeature.txt.
   - *User Initials*: Type a maximum of three characters to identify the creator of the contact. The initials are used in the editor as part of a contact identifier.
   - *Prefix*: The prefix is usually the line number. It is combined with the contact number to generate a 12-character key for database applications.
   - *Image Width*: (optional) Enter a width for any image file created from the contact.

- *Image Length*: (optional) Enter a length for any image file created from the contact.
- *Display with single colour*: Select this to have all contact markers and their identifiers appear in the same default colour. (The default colour can be set in Side Scan Editor Display Options on the Tools menu.) Leave blank to have a one colour apply to the contact markers and a different colour applied to the bounding box and contact labels.

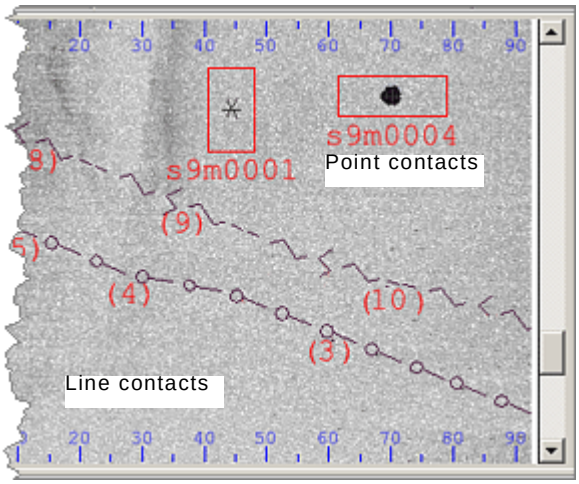
Point Contact

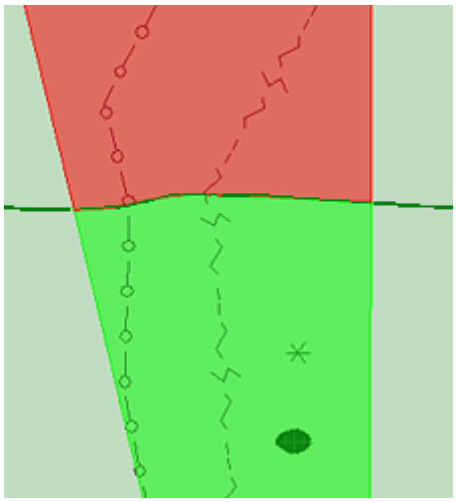
Menu	Tools > Side Scan Editor > Contacts > Add Point Contact
Tool	

3. To insert a point contact, select the Add Point Contact tool and click where you want to place the contact.

A marker is placed in the Side Scan Editor at the point where you clicked.

The type of contact is indicated by the symbol displayed. The contact is identified by the initials entered in the General tab, and a four-digit number (starting at 0001). The information entered in the General tab is assigned to the contact.





The image bounding box always snaps to the contact so that the contact point is in the middle of the box. The bounding box represents the image area of the contact. You can adjust the size of the image using the *Image Width* and *Image Length* fields on the Contract Details tab in the Control window.

Line Contact

Menu	Tools > Side Scan Editor > Contacts > Add Line Contact
Tool	

To insert a contact record for a line:

1. Select the Add Line Contact tool and click the point where you want to insert the contact.
2. Click another section of the Side Scan Editor.

The two points are joined by a line.

3. Continue adding line segments as needed to represent the contact.

4. Select **Edit Line**> **Remove last** to delete the line segment just created.
5. To escape the Line Contact tool, right-click the mouse and select **Edit Line**> **End** from the pop-up menu.

The contact lines are drawn in the Side Scan Editor. Each line is identified with a number, and each type of line is drawn in a different style. The information entered in the General tab is applied to the contact. The contact is identified with the initials entered in the General tab, along with a four-digit number (starting at 0001).

Repeat New Feature

When the Repeat New Feature tool is activated, you can add any number of contacts of the same type and attributes, without having to set each one separately.

To add multiple similar contacts:

Menu	Tools > Side Scan Editor > Contacts > Repeat New Feature
Tool	

1. Select the command.
2. Enter contact information in the appropriate fields on the General tab in the Control window.
3. Use the Add Point Contact tool to place the contacts, one after the other, into the Side Scan Editor.

The Repeat New Feature tool button will remain depressed while you add contacts. To change to a different type of contact or to change contact data, deactivate the Repeat New Feature.

Editing Contacts

You can reposition both line and point contacts and update the data associated with a contact.

The Select Contact tool is used to select point or line contact(s) for editing.

Select Contact

Menu	Tools > Side Scan Editor > Contacts > Select Contact
Tool	

1. Choose the Select Contact tool.
2. To select a *single* contact, click on or draw a box around the contact so that it becomes highlighted. Clicking on any part of a line will select the entire contact.
3. To select a number of contacts, press the mouse button and drag the cursor across an area so that the contacts within it are highlighted.

Move Contact

Menu	Tools > Side Scan Editor > Contacts > Move Contact
Tool	

1. To move a selected contact, select the Move Contact tool.
2. Position the cursor on top of the selected contact. Press and hold the mouse button. This will turn the highlighted contact into a red box with direction arrows.
3. Drag the contact to a new position.
4. Deactivate the Move Contact tool to finalize the new position of this contact. You are returned to the Select Contact tool.

You can use the Move Contact tool to move an entire line contact, or to adjust or realign part of the line.

5. To move a selected line contact, position the cursor on any part of the line, press and hold the mouse button, and drag the whole line to a new position.
6. To realign *part* of a selected line, click on the line. This will highlight nodes which will appear as squares on the line.
7. Click on the node to be adjusted, which will turn the node square red. Position the cursor on the node. Press and hold the mouse button, and drag it to a new position. This will pull the attached lines with it.

Changing Contact Information

You can change the information that is associated with the contact when it was first created, either by typing it in or with measurement tools.

1. Choose the Select Contact tool.
2. Select a single contact.

The contact is highlighted and the contact information is displayed in Contact Details tab in the Control window.

3. Type new information in the fields and/or select a new object type from the drop-down list.
4. Click **Update**.

Use the Measure Distance and Measure Shadow tools on the Side Scan Editor toolbar to amend the contact information.

Use Measure tools

5. To update the contact width information, choose the Measure Distance tool and press and hold <Ctrl> while dragging the cursor across the area of the contact to be measured.

If measuring in the Side Scan Editor window, the value in the Contact Width field will reflect the measurement as you are make it. If measuring in the Zoom tool, the value will be updated when the mouse button is released.

6. To retain the updated information, release the mouse button and then the <Ctrl> key.
7. To update the Contact Length information, choose the Measure Distance tool and press <Alt> while dragging the cursor across the contact area.
8. To update the Contact Height field, select the Measure Shadow tool and press <Ctrl> while dragging the cursor across the contact area.
9. Click **Update**. If you do not click Update, the values will be lost when the contact is de-selected.

Menu	Tools > Side Scan Editor > Measure Shadow
Tool	

The contact information is updated.

Set contact status

You can also use the Select Contact tool with the Query, Reject or Accept tools. A rejected contact can deleted from the track line using the Purge Rejected Contacts command.

10. Select either an Accept, Reject or Query tool (see "EDIT SIDE SCAN DATA" ON PAGE 510).

- If you selected the Query tool, data relating to that contact is displayed in the Query tab of the Worksheet window.
- If you selected Accept or Reject, the contact is assigned the appropriate status flag.

Purge Rejected Contacts

11. To delete a rejected contact, select the contact (using the Select Contact tool) and select the Purge Rejected Contacts command.

Menu	Tools > Side Scan Editor > Contacts > Purge Rejected Contacts
Tool	

Recompute Contact Position

Update the geographic position of contacts after towfish navigation has been recomputed for a track line or group of track lines (see "SLANT RANGE CORRECTION" ON PAGE 525).

1. Select the ship or towfish track lines layer in the Session tab of the Control window.
2. Select one or more track lines.
3. Select the command.

Menu	Process > Recompute Contact Position

Position information for contacts along the selected track lines are updated to match the new towfish navigation.

The start time, end time and elapsed time for the recompute process is reported in the Output window. Any error in the processing will be reported there also.

View contacts while scrolling

If you want your contacts to remain visible in the waterfall display while scrolling along a track line or using the Playback functions

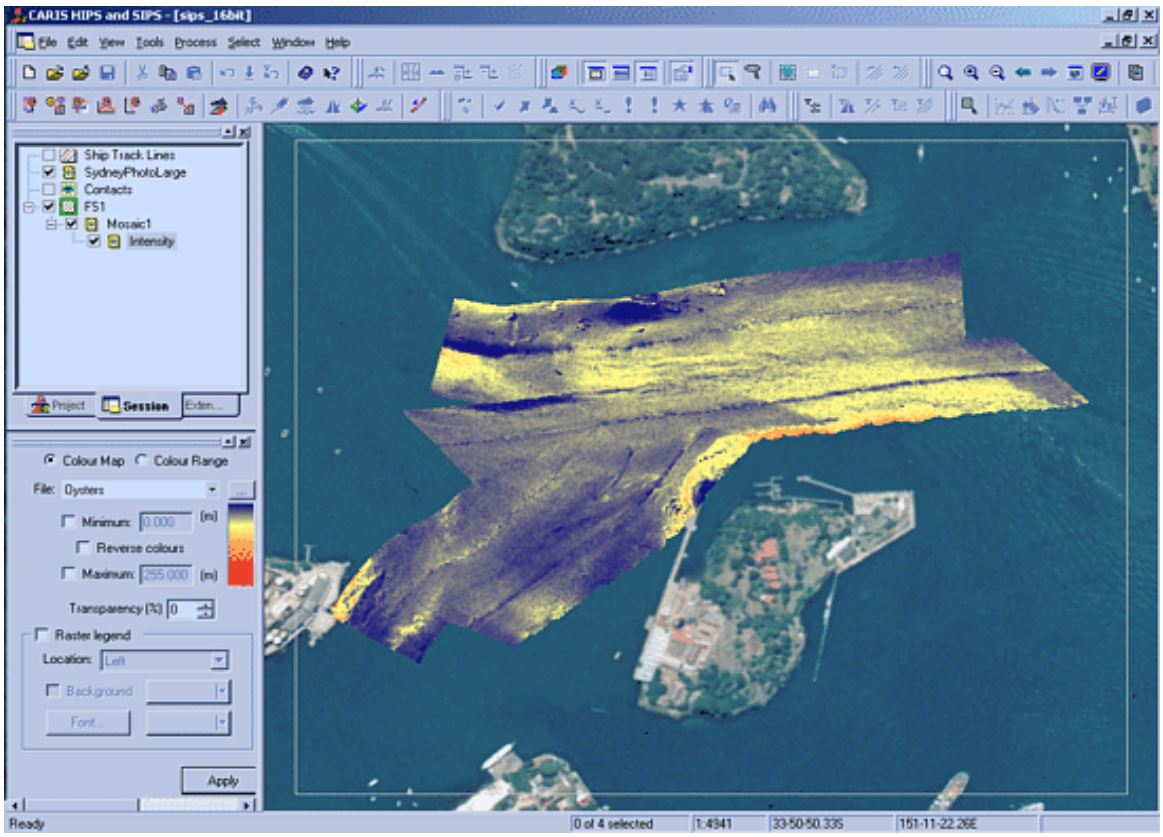
1. Select Side Scan Editor from the Tools > Options > Display tab.
2. Select the *Display contacts during playback* check box, and click Apply.

Mosaics

A mosaic is a georeferenced TIFF image file created from one or more track lines—within a field sheet—that contains slant range corrected side scan or amplitude data.

The mosaic process combines imagery data with heading and navigation data to georeference the sonar pings within the geographic extent of the field sheet.

Below is an example of finished mosaic in the Display window.



Mosaics are displayed as layers in the Session tab of the Control window:

The mosaic layer is named according to the name given in the Mosaic Wizard. The mosaic layer also contains a Intensity sub-layer. When selected this layer displays a screen tip showing the intensity values for each section of the mosaic when the cursor is moved over the mosaic

Create Mosaics

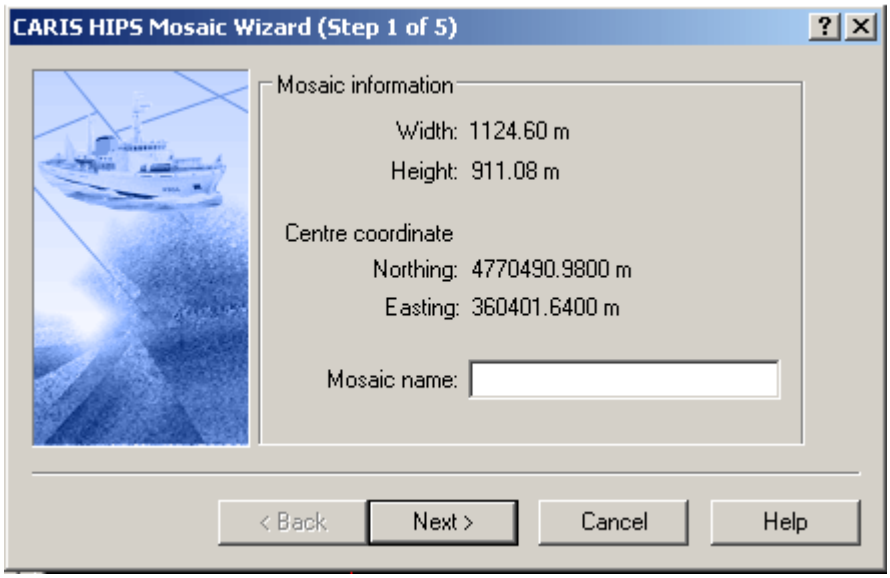
A mosaic is created as a new field sheet layer using the Mosaic Wizard.

1. Make sure a field sheet has been created for the area to be mosaicked (see "CREATE A FIELD SHEET" ON PAGE 349)
2. Select the field sheet file in the Control window.
3. Select the New Mosaic tool.

Menu	Process > Mosaic > New…
Tool	
Pop-up	New Mosaic

Create Mosaic - Step 1

The Mosaic Wizard (Step 1) dialog box is displayed.

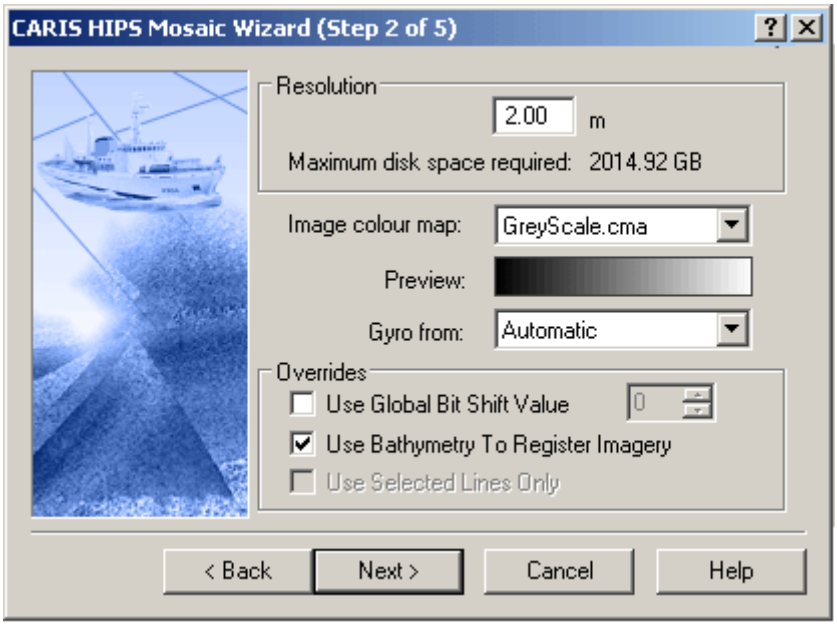


The width, height and centre coordinates are derived from the field sheet.

4. Type a name for the mosaic.
5. Click **Next**.

Create Mosaic - Step 2

The Mosaic Wizard (Step 2) dialog box is displayed.



6. Enter the pixel resolution. For example, if the resolution is 2.00 metres then one pixel represents two metres. The amount of space needed to store the mosaic data is shown below the Resolution box. The smaller the resolution value, the greater the space needed to store the data.
7. Select a colour map from the drop-down list. A preview of the colour map is provided below the list.

You can pick what orientation source to use from the drop-down list. If you select *Automatic*, SIPS uses this order of precedence:

- SSS Gyro
- Ship Gyro
- Calculated CMG from SSS Navigation
- Calculated CMG from Ship Navigation.

8. Select an orientation source in the *Gyro from* drop-down list.

When 16-bit data is opened in the Side Scan Editor, the bit shift value is saved in SIPS. This means that different track lines might have different bit-shift values and this could affect the mosaic display. The *Use Global Bit Shift Value* option applies the same bit-shift value to all lines.

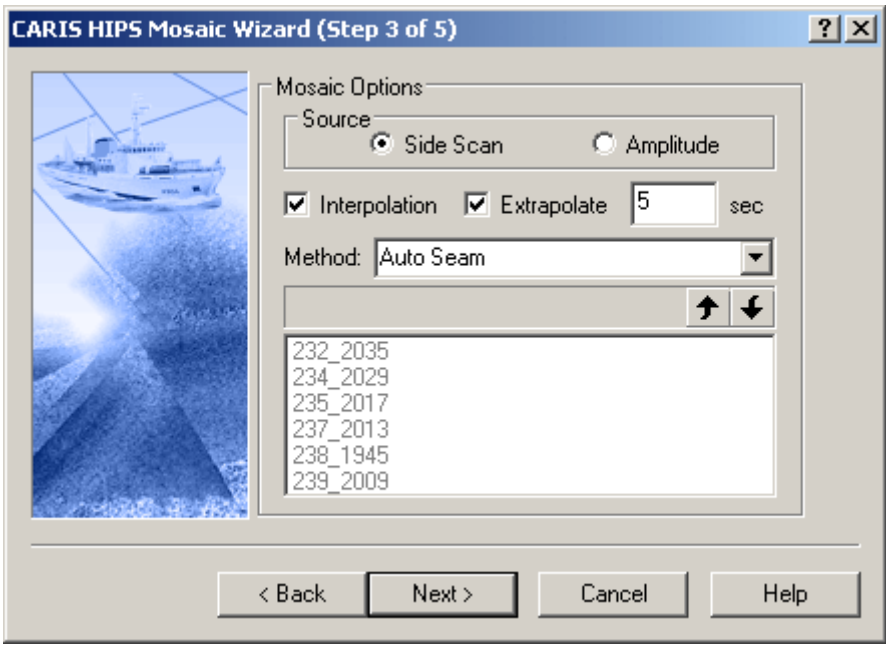
9. To apply a uniform bit-shift value to 16-bit data, select the *Use Global Bit Shift Value* check box and then select a value in the associated field.

If the data contains bathymetry as well as side scan data, you can use the information from the depth file to position the side scan imagery for the mosaic

10. Select *Use Bathymetry To Register Imagery* to position the side scan imagery.
11. If you have track lines selected, the *Use Selected Lines Only* check box is selected by default. If you do not want to use only the selected lines, remove the check mark.
12. Click **Next**.

Create Mosaic - Step 3

The Mosaic Wizard (Step 3) dialog box is displayed.



1. Select either *Side Scan* or *Amplitude* to determine the data source for creating the mosaic.
2. Select *Interpolation* if you want to fill in any gaps between the sonar pings.

Sonar, heading and navigation data readings are recorded at different intervals. Therefore, sonar pings at the start and end of the track line may be excluded from the mosaic because there is no heading and/or navigation data to interpolate. The *Extrapolate* option extends the heading and/or navigation data at each end of the line to a maximum specified by the selected value.

3. Select the *Extrapolate* check box to enable this option.
4. Type a time range (in seconds) to extend heading/navigation to sonar data within the selected range.

The *Method* drop down list determines what pixel values are selected for mosaic creation.

5. Select from the following options in the *Method* drop-down list:
    - Auto Seam: Determine which pixels have higher significance when pings overlap. A weighting function is constructed for each ping that assigns lower significance to pixels at the edge of the ping than to pixels near the centre. The first few pixels near the nadir are assigned low significance due to the low resolution in that region.

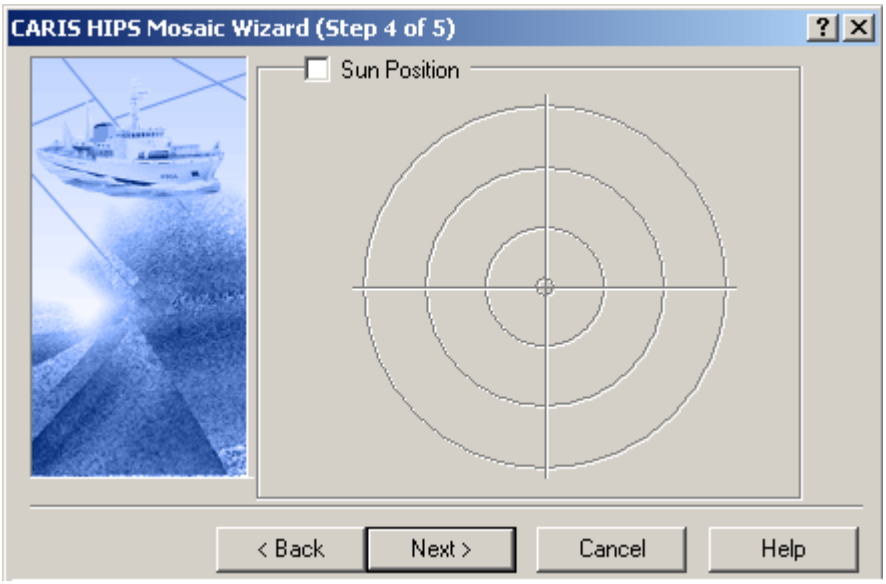
- Shine Thru: Use only the highest-weighted pixels from the overlapping track lines.
- Overwrite: Imagery from each successive line is added to the mosaic to overwrite the previous data.
- Individual Mosaics: Create single mosaics for each track line without overlap from other lines.

When the *Overwrite* option is selected, a list of track lines in the field sheet is active. This list determines the order that the lines are processed and also determines the order that data is overwritten when track lines overlap.

6. Change the track line processing order by selecting a line and clicking the up or down arrow buttons.
7. Click **Next**.

Create Mosaic - Step 4

The Mosaic Wizard (Step 4) dialog box is displayed.



The *Sun Position* option determines the direction of the shadow in the mosaic. Either port or starboard imagery will not be used in the mosaic to display shadows correctly. Two-hundred percent coverage is recommended to reduce gaps in the mosaic. If the Sun Position option is not checked all imagery is used in the mosaic.

8. Make sure the *Sun Position* check box is selected to use the lighting compass.
9. Place the cursor over the icon, press and hold the mouse button, and drag the sun icon to the desired position.
10. Click **Next**.

Create Mosaic - Step 5

The Mosaic Wizard (Step 5) dialog box is displayed.

This dialog box applies filtering options to exclude mosaic data based on across-track distance and/or altitude ratios. The options can be applied to the inner part of the mosaic (nadir) and/or to the edges (outside edge).

The *Min Acrosstrack/Altitude Ratio* excludes data starting from nadir extending across-track to a distance determined by the towfish altitude and the given ratio.

The *Min Acrosstrack Distance* option can also be selected along with this option.

1. Select the *Min Acrosstrack/Altitude Ratio* check box to enable this option.
2. Type the ratio value in the field.

If the *Min Acrosstrack/Altitude Ratio* option is selected, then the *Min Altitude Limit for Ratio* option is available. This option sets a minimum fish altitude for the M*in Acrosstrack/Altitude Ratio*. For fish altitudes less than this limit, the Min Acrosstrack/ Altitude Ratio is NOT applied.

1. Select the *Min Altitude Limit for Ratio* option, if needed.
2. Type the minimum altitude limit.
3. Select the *Min Acrosstrack Distance* option to use the minimum distance from the centre of the side scan ping.

4. Type the across-track distance (from nadir) for the mosaic in the appropriate field.

The *Max Acrosstrack/Altitude Ratio* excludes data starting from the distance determined by the towfish altitude and the given ratio to the outside edge.

The *Max Acrosstrack Distance* option can also be selected along with this option.

1. Select the Max Acrosstrack/Altitude Ratio check box to enable this option.
2. Type the ratio value in the field.

If the *Max Acrosstrack/Altitude Ratio* option is selected, then the *Max Altitude Limit for Ratio* option is available. This option sets a maximum fish altitude for the *Max Acrosstrack/Altitude Ratio*. For fish altitudes greater than this limit, the Max Acrosstrack/Altitude Ratio is NOT applied

1. Select the Max Altitude Limit for Ratio option, if needed.
2. Type the maximum altitude limit.
3. Select the Max Acrosstrack Distance option to use the maximum distance from the centre of the side scan ping.
4. Type the maximum across-track distance for the mosaic in the appropriate field.
5. Click **Finish**.

The status bar shows the progress of the mosaic-building process.

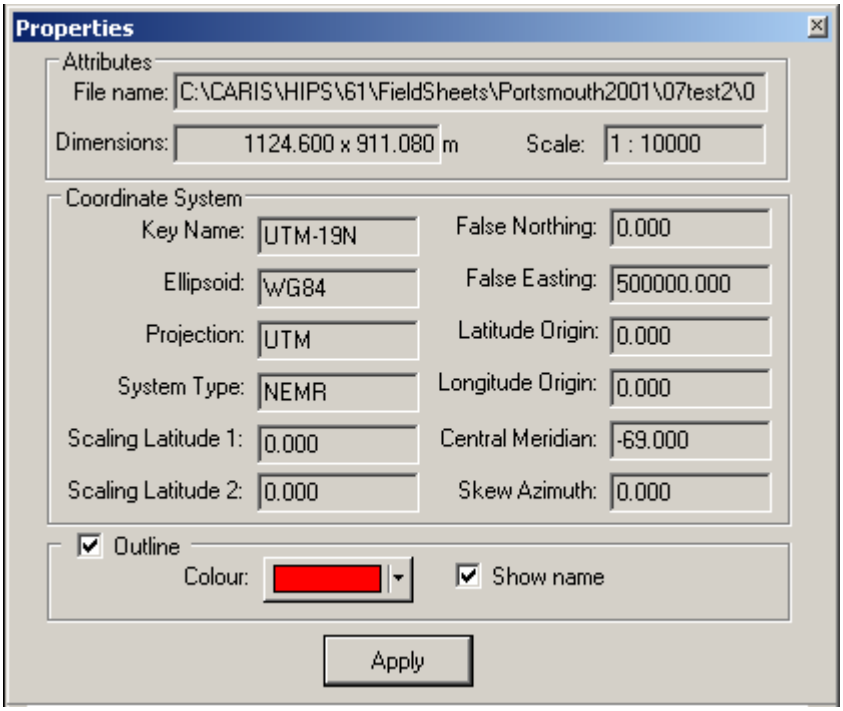
Mosaic Attributes

View the attributes of a mosaic.

1. Select the mosaic layer in the Session tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up window.

Menu	Window > Properties
Tool	
Pop-up	Properties

The *File Name*, *Dimensions, Scale,* and coordinates fields are displayed in the window. These are read-only and cannot be modified.



At the bottom of the Properties window you change the colour of the mosaic outline. You can also turn off the file name so that it does not show in the mosaic display.

Intensity Properties

Adjust colour settings using the Properties window.

1. Select the Intensity sub layer in the Session tab of the Control window.
2. Select the Properties window command, or right-click the layer and select Properties from the pop-up window.

The display options for the mosaic are identical to the BASE Surface display properties, see "SET ATTRIBUTE PROPERTIES" ON PAGE 414 for more information.

Add To Mosaic

Add a selected track line to an existing mosaic.

1. While a mosaic is open in the Display window, select Track Lines layer in the Control window.
2. Select the track line you want to add.

Add to and Remove From Mosaic

3. Right-click the Mosaic layer in the Control window, and then select the Add to tool.

Pop-up	Add to

The Step 3, Step 4, and Step 5 dialog boxes from the Mosaic wizard are displayed in sequence (see "CREATE MOSAICS" ON PAGE 546).

4. Enter data as needed.
5. Click **Next** to move to the next dialog box.
6. Click **Finish** to regenerate the mosaic (Step 5).

The mosaic is regenerated to display the new line.

Remove From Mosaic

Remove a selected track line from an existing mosaic.

1. While a mosaic is open in the Display window, select Track Lines layer in the Control window.
2. Select the track line you want to add.
3. Right-click the Mosaic layer in the Control window and select the Remove from Mosaic command.

Pop-up	Remove From

The selected track line is removed from the mosaic. The mosaic is flagged as outdated and must be regenerated.

Recompute

Regenerate a mosaic that has been flagged as outdated.

1. In the Session tab of the Control window, select a mosaic that has been flagged as outdated.
2. Right-click the Mosaic layer in the Control window and select the command.

Pop-up	Recompute

The mosaic will be regenerated. The mosaic is no longer flagged as outdated.

Image Classification

Image classification groups return-signal intensities (represented in the mosaic as pixel values) into classes. These classes are displayed in the mosaic using specific colours.

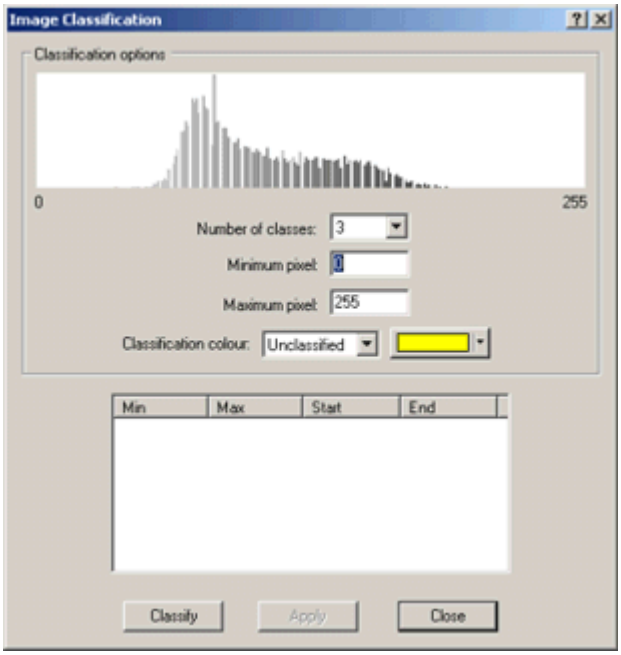
Image classification

Prior to setting image classification, you may need to examine the mosaic to determine the number of classes needed to represent pixel values.

1. [Optional] Select the Intensity sublayer of the mosaic in the Session tab and move the cursor across the mosaic in the Display window. As you move the cursor, screen tips show the pixel value.
2. Select the Classify command.

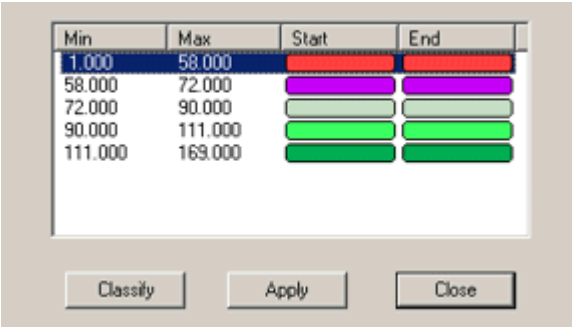
Menu	Process > Mosaic > Classify

The Image Classification dialog box is displayed.

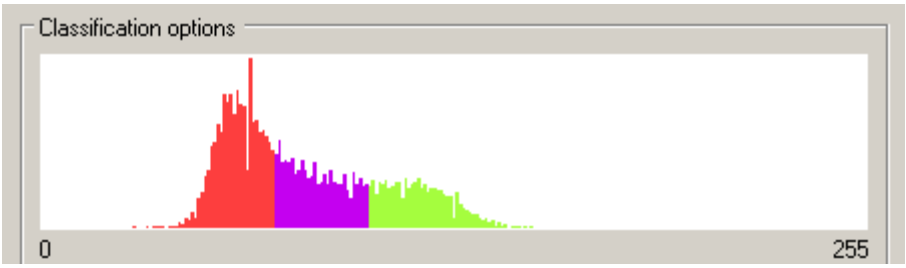


3. Use the *Number of classes* drop-down list to set the total classes in the mosaic.
4. Set a range of pixel values for a class by typing *Minimum pixel* and *Maximum pixel* in their respective fields.
5. Select a colour for the class by selecting a class from the *Classification* colour drop-down list. A colour is automatically displayed beside the class name.
6. [Optional] Select another colour from the colour picker, or create a custom colour from the standard Windows colour palette.
7. Repeat Steps 4 to 7, as needed.
8. Click **Classify**.

A colour table is generated according to the number of classes you selected in Step 3. Pixels are grouped according to class starting from the minimum to the maximum pixel value.



The histogram is refreshed to show the pixels coloured according to the class designation



.

9. Click **Apply** to save the classification system to a .CRF (colour range file).

A Save As dialog displayed. The selected colours can be saved as a colour map for future use.

10. Type a name for the colour map and click **Save**.

The colours are displayed in the mosaic.

SIPS Template Wizard

The SIPS Template Wizard combines a number of side-scan correction tools into a single wizard. The SIPS Template Wizard can be used with the Side Scan Editor open to process a single line or with the Editor closed to process one or more selected track lines.

The following operations can be applied using the Template Wizard:

- "Slant Range Correction" on page 525
- "Beam Pattern Correction" on page 529
- "Angle-Varying Gain" on page 534
- "Time-Varying Gain" on page 535
- "Despeckle" on page 538

The settings entered into the wizard can be saved to a SIPS Template File (STF) that can be loaded and used again.

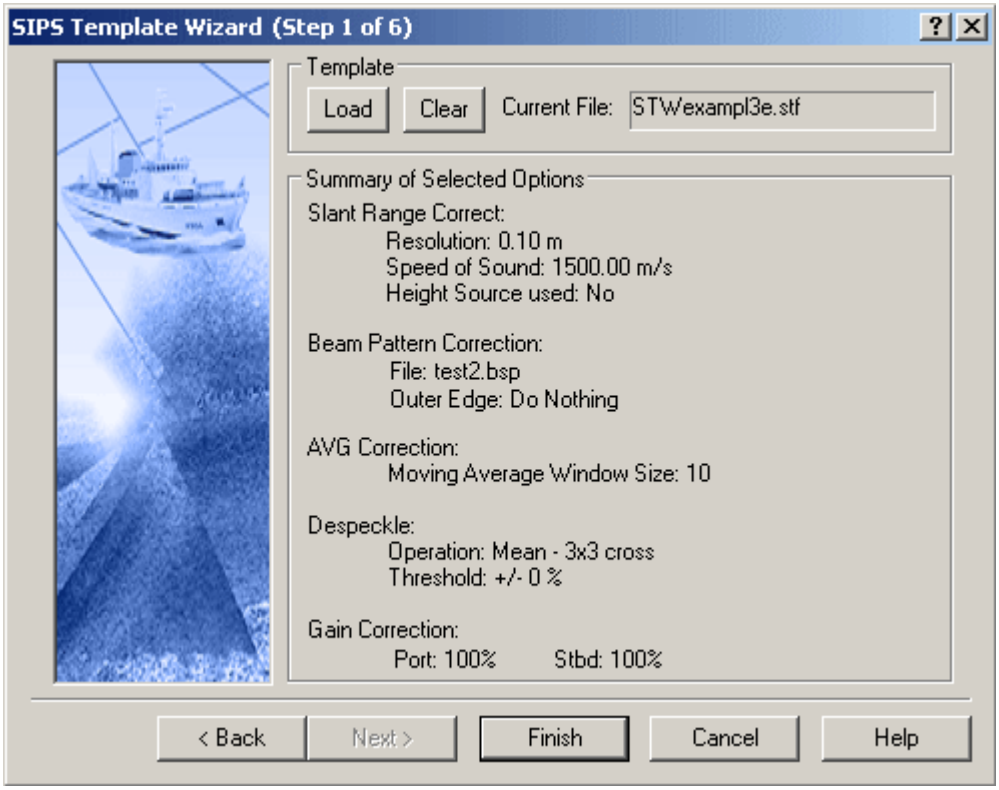
1. Select one or more track lines.
2. Select the tool.

Menu	Process > SIPS Template Wizard
Tool	

SIPS Template Wizard (Step 1)

The SIPS Template Wizard (Step 1) dialog box is opened so you can create a template or load an existing one.

The example below displays the options and values for a loaded SIPS Template File.



To create a new SIPS template file, click **Next**.

To load an existing SIPS Template file:

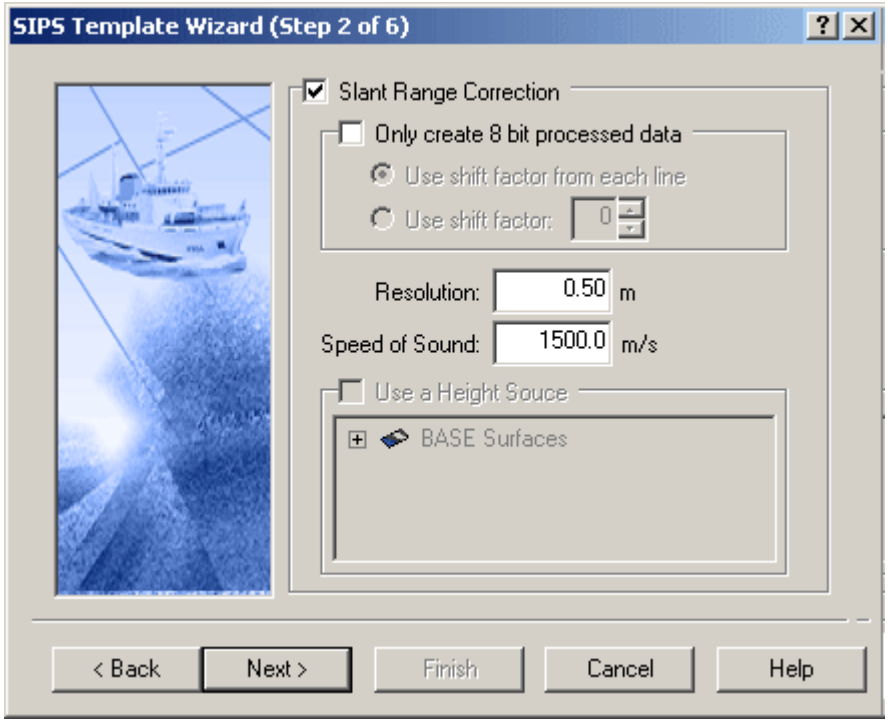
1. Click **Load** and choose an existing file. The file name is displayed in the Current File field. The Summary provides an overview of the settings.
2. To remove the selected file, click **Clear.**
3. To execute the loaded SIPS Template file, click **Finish**.

This will launch the SIPS Template Executive to execute the wizard options.

4. Click **Process** to execute the corrections selected in the SIPS Template Wizard.
5. If you want to modify the loaded options, click **Next**.

SIPS Template Wizard (Step 2)

The SIPS Template Wizard (Step 2) page of the Wizard displays all options available for Slant Range Correction.



1. Select *Slant Range Correction* to enable this correction process.
2. To apply an existing bit factor, select the *Use Shift Factor From Each Line* option.
3. To apply a new bit factor to the data, select the *Use Shift Factor* option and select the bit factor by clicking the arrow buttons beside the bit factor size box.
4. Type the *Resolution* for the slant range corrected file.
5. Type the average speed of sound.

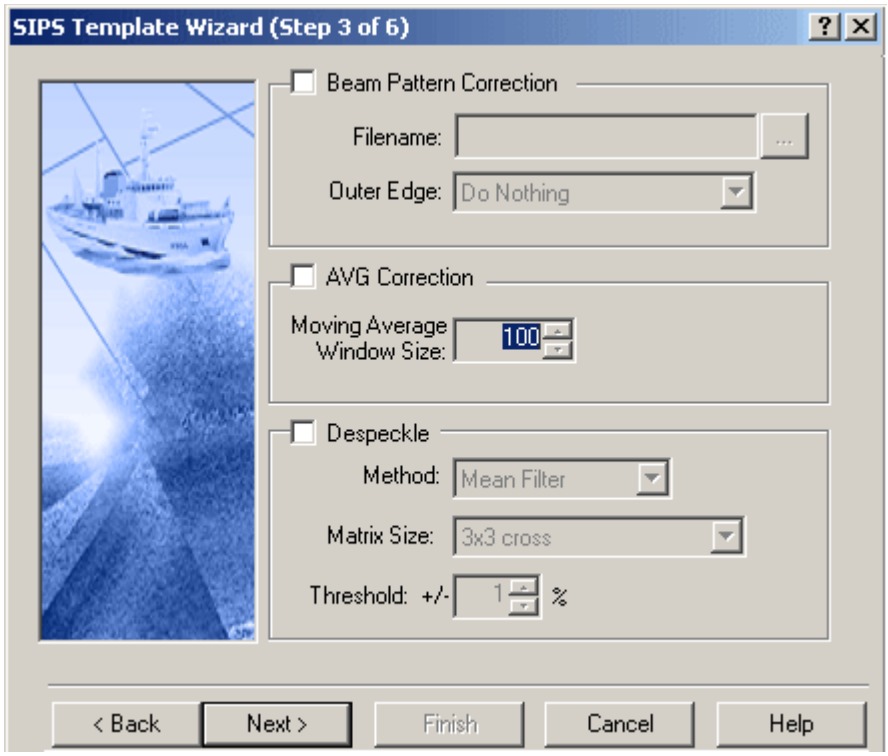
A height source (BASE Surface, grid, or tile) can be used to perform slant range correction. This can improve the across-track positioning of pixels especially over varied or steeply sloping terrain.

6. Select the *Use Height Source* check box to implement this option, and select the height source from the list below.
7. Click **Next**.

For more information, see "SLANT RANGE CORRECTION" ON PAGE 525.

SIPS Template Wizard (Step 3)

The SIPS Template Wizard (Step 3) dialog box contains the options for applying corrections to sonar data.



Beam Pattern Correction removes along-track banding.

1. Select the *Beam Pattern Correction* check box to enable this option and browse to select an existing BPC file.

The outer edges of some pings may have been recorded at angles beyond the extent of the beam pattern file being used. The Outer Edge option must be specified for handling these cases.

2. Select a method for handling sonar data collected outside the bound of the beam pattern file:
   - Extrapolate: Extrapolate the beam pattern to correct angles beyond the outer edge of the generated beam pattern file.
   - Trim: Remove sonar data that is beyond the extent of the beam pattern file. Trimmed data will appear as 0 intensity white space in the Side Scan Editor.
   - Do Nothing: Does not correct sonar data beyond the extent of the beam pattern file. Uncorrected data will appear the same before and after Beam Pattern Correction in the Side Scan Editor.

Angle-Varying Gain (AVG) removes the angular response of sediment from the imagery, normalizing the mean angular intensities, with a moving average filter applied ping by ping to selected areas of data.

3. Select the *AVG Correction* check box to enable the process.
4. Select the number of adjacent pings to include in the moving average by clicking *Moving Average Window Size* up or down arrow buttons.

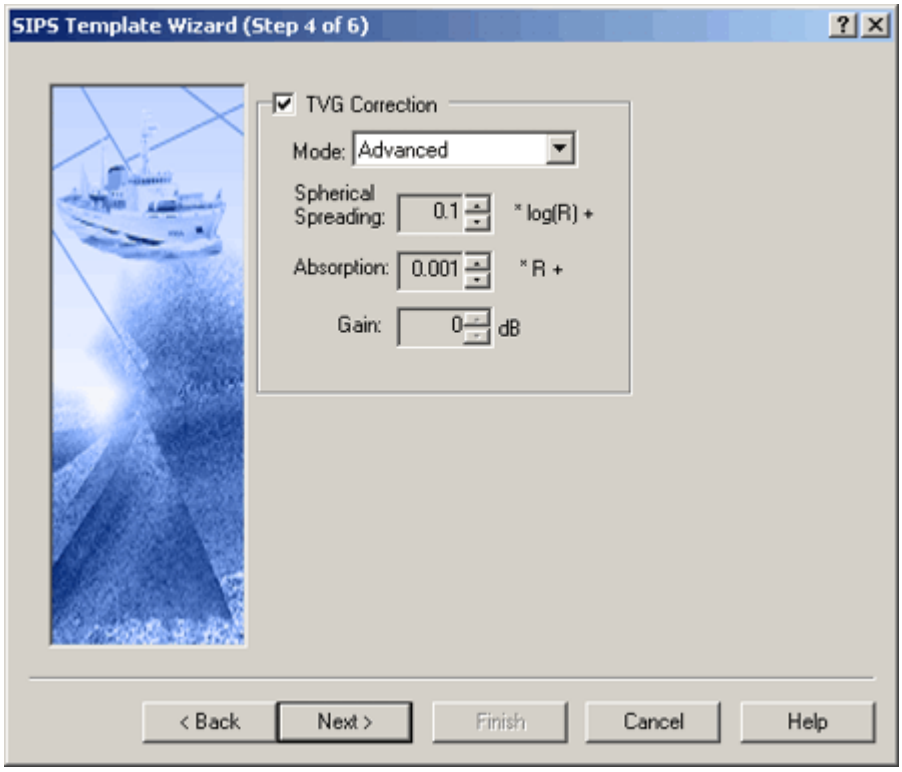
The Despeckle function removes "noise" in the processed side scan image. (Noise consists of randomly scattered pixels that are brighter or darker than their neighbouring pixels).

5. Select a *Method* from the drop-down list.
6. Select a *Matrix Size* from the list
7. Specify a *Threshold Factor* in the box.
8. Click **Next**.

For further information, see "Beam Pattern Correction" on page 529, "Angle-Varying Gain" on page 534 or "Despeckle" on page 538.

SIPS Template Wizard (Step 4)

The SIPS Template Wizard (Step 4) dialog box contains options for applying the time-varying gain correction.

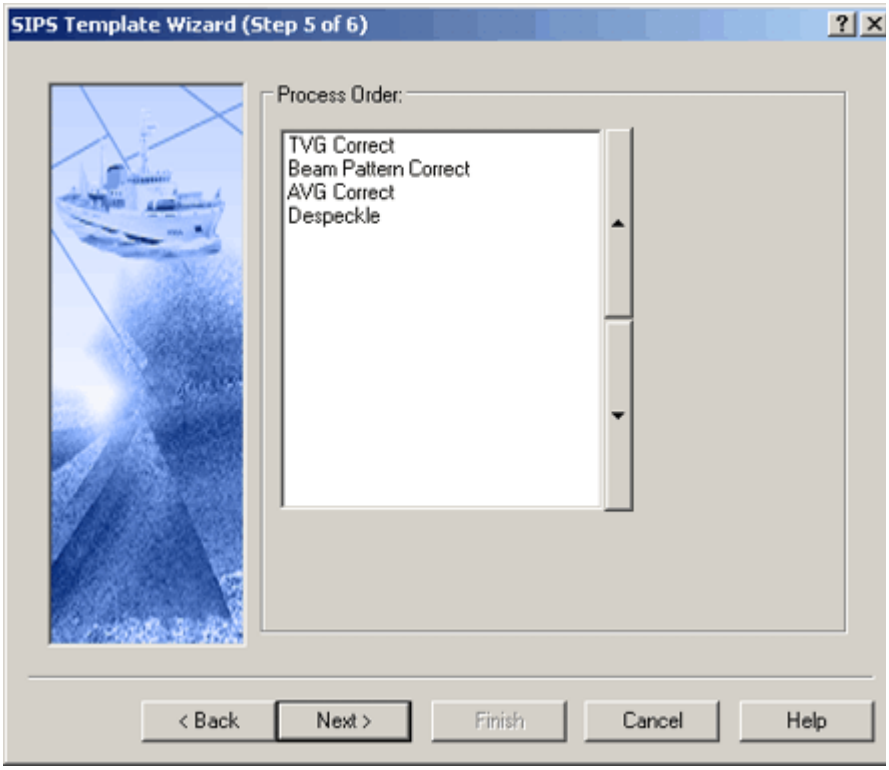


Time-Varying Gain Correction adjusts signal intensities by applying a non-uniform, time-dependent gain.

1. Select the *TVG Correction* check box to make this option active
2. Select the Standard or Advanced modes from the *Mode* drop-down list.
3. For further information, see "TIME-VARYING GAIN" ON PAGE 535.
4. Click **Next**.

SIPS Template Wizard (Step 5)

The SIPS Template Wizard (Step 5) dialog box lists the order in which the options will be processed on the track line(s), and allows for this order to be adjusted.

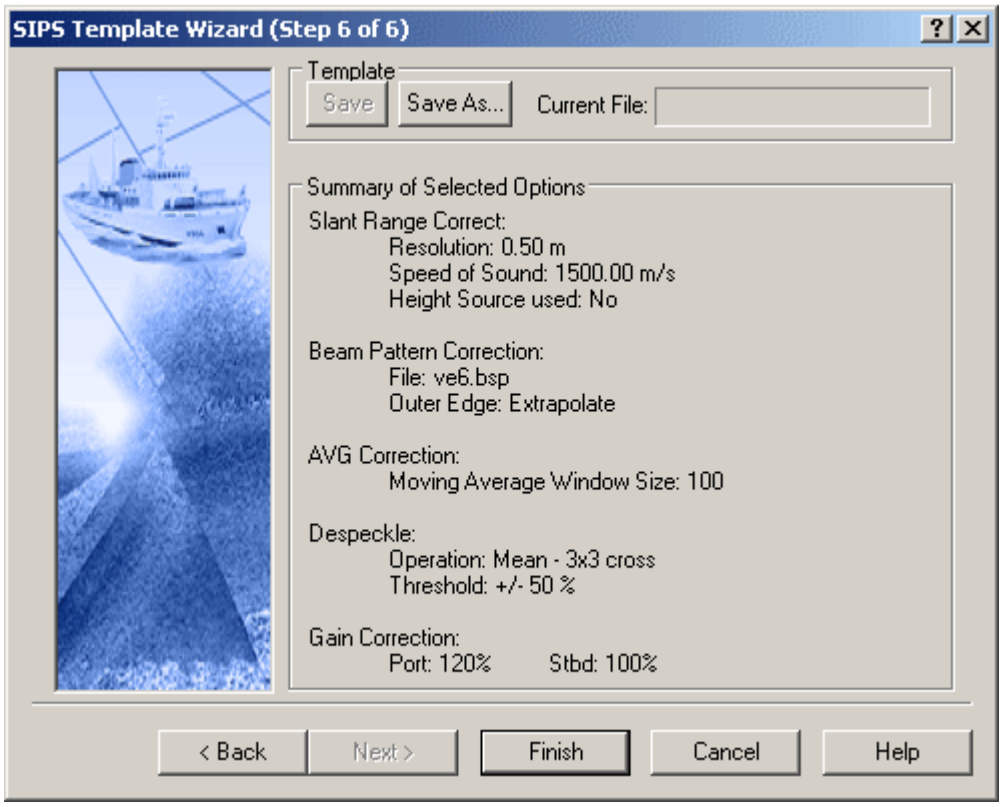


The *Process Order* list contains all options selected in the previous wizard dialog boxes in the order that the operations will be processed on the track line.

1. If you want to change the processing order, select a correction and use the Up or Down arrow buttons to change its position in the list. The item at the top will be executed first.
2. If selected, Slant Range Correction will be executed first. The order of this correction cannot be changed.
3. Click **Next**.

SIPS Template Wizard (Step 6)

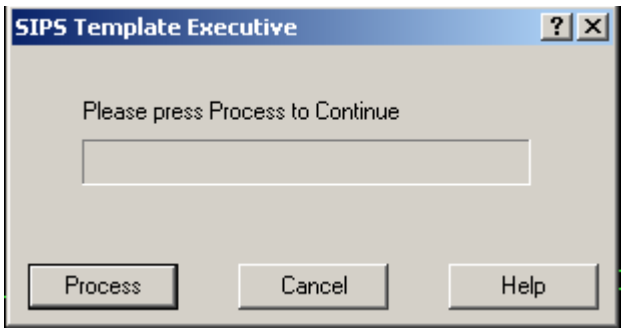
The SIPS Template Wizard (Step 6) displays the options and associated values selected in the previous dialog boxes.



The options you have selected in the previous dialog boxes are displayed in the *Summary of Selected Options* section. Review the summary before closing the wizard.

1. If needed, click **Back** to get a previous dialog box to make changes.
2. If you wish to save this as a template, click **Save** to overwrite the currently loaded template file, or click **Save As** to create a new template file with these options.
3. If you do not wish to execute your template at this time, click **Cancel** to close the wizard. If you have saved your template, then your selection options will not be lost.
4. To immediately execute the options selected in the Wizard, click **Finish.**

This will launch the SIPS Template Executive to execute the wizard options.



5. Click **Process** to execute the corrections selected in the SIPS Template Wizard.
6. Once processing has commenced, clicking the **Cancel** button will not immediately terminate the Executive. The Executive will close once the current correction is finished.

For example, if you have selected four corrections and you press cancel during second correction, then the Executive will terminate at the end of the second correction.

SIPS Template Wizard can also be also activated from the task list in the Batch Processor.

23

Batch Processing

Overview

Batch processing automates processing by combining a number of tasks (Tide Correction and SVP Correction, for example) into a single batch file that can reopened and used on multiple track lines.

The batch processing function is available in all licensing levels of HIPS and SIPS, but the processing options available are determined by the processing functions contained with the license.

All data entered into the batch processor is saved in a HIPS Batch Processing File (HPB) file. HPB files are in XML format and can be viewed in any text editor or XML-compliant browser. Batch processing files are saved in ..\Hips\Session and can only be used in the Batch Processor.

HIPS filter files (HFF) can also be opened in filtering options of the Batch Processor.

The Batch Processor is a separate application that is opened from the HIPS interface.

Open Batch Processor

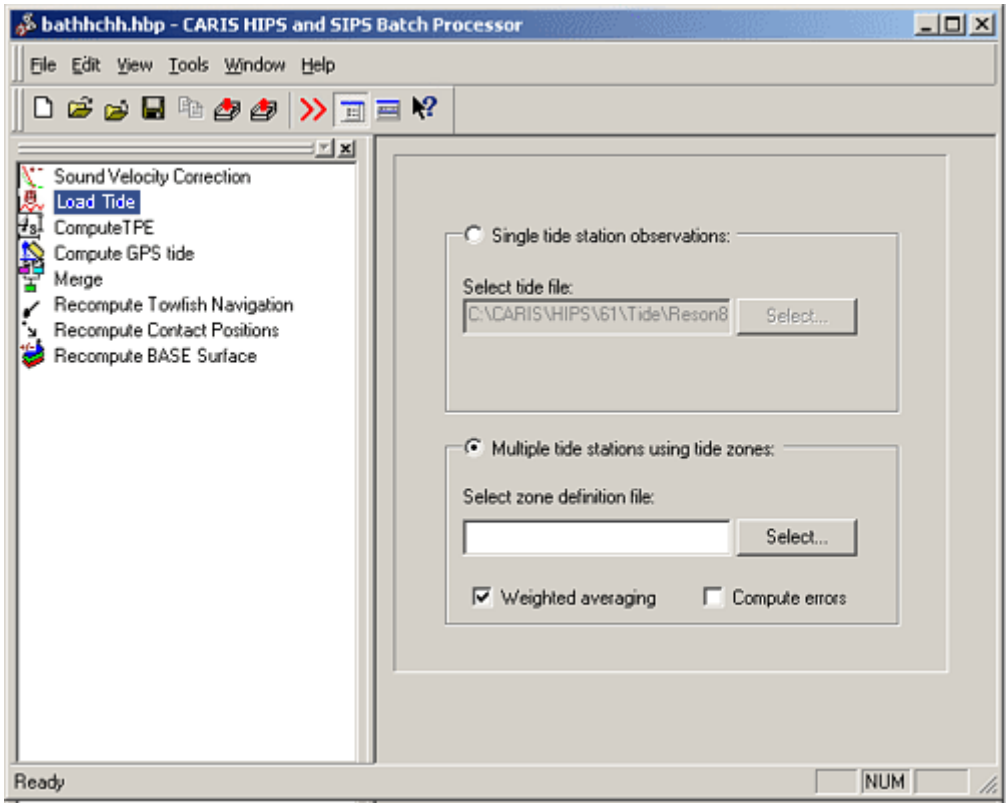
Batch Processor

Menu	Tools > Batch Processor
Tool	

1. Select the Batch Processor command. A project or session file does not have to open in HIPS and SIPS.

The Batch Processor is opened.

The following image shows the Batch Processor with a complete list of tasks.



The Batch Processor contains three windows.

- The window on the left contains a list of tasks that are open in the processor.
- The window on the right displays the options associated with a task.
- The window on the bottom displays the results of the processing operation.

When the processor is first opened, it ready to create a new HBP file.

Running the Batch Processor

The following sections outline the workflow for opening or creating a HPB file and running the HIPS Batch Processor.

Open HBP File

Open an existing HBP file.

1. In the Batch Processor interface, select the Open File command.

Menu	File > Open
Tool	
Key	<Ctrl+O>

The Open dialog box is displayed.

2. Select a file and click **Open**.

The file is displayed in the Batch Processor.

New HBP File

Create a new HPB file.

When the Batch Processor is first opened, it is ready to create a new HPB file. If a HPB file is already open and you select the New File command, the Processor is reset for the new file (the already open file are is closed).

New File

1. Select the New File command.

Menu	File > New
Tool	
Key	<Ctrl+N>

A new file is ready to be modified in the Batch Processor.

Save/Save As

1. In the Batch Processor interface, select a Save command to save a HBP file for the first time or save an existing file under a new name.

Menu	File > Save/Save As
Tool	
Key	<Ctrl+S>

All files are saved to …\Hips\Session.

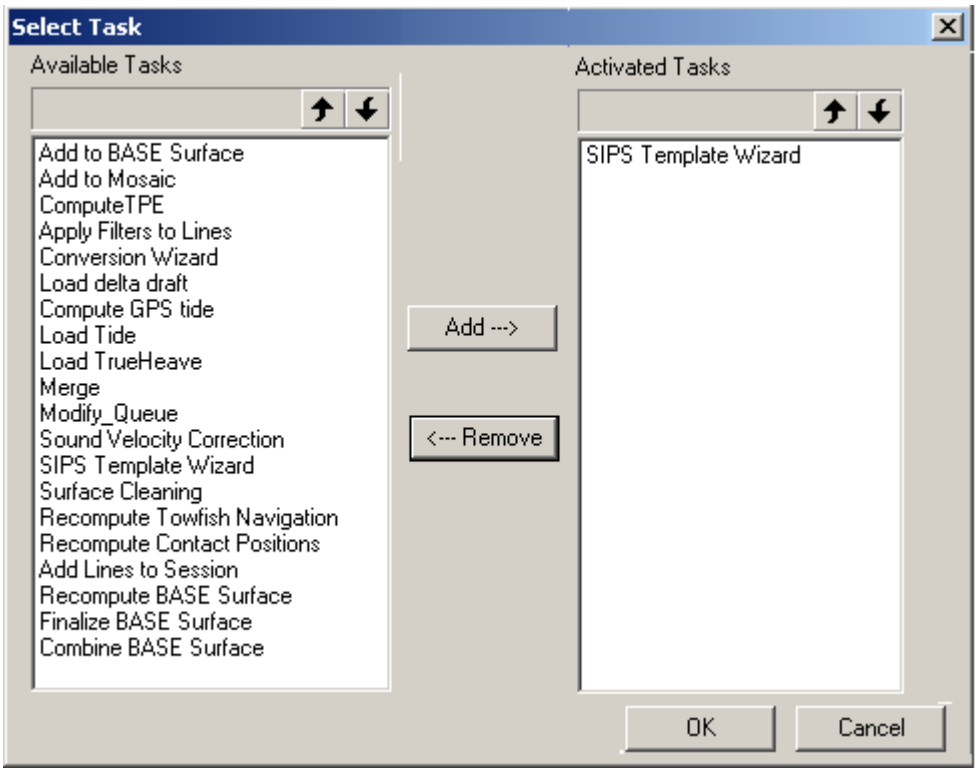
Select Process

A process is a specific order of tasks that are run through the Batch Processor. Each process is saved to the current HBP file.

Select the Add Process command.

The Select Task dialog box is displayed.

> Remember that the list of tasks in Activated Tasks must be in the same order as the workflow in HIPS and SIPS: the Compute TPE task must be listed before the Merge task, and so on.



2. Select tasks for the process in three ways:
   - Select an individual task so it is highlighted.
   - Select a group of tasks by holding the <Ctrl> key and clicking the required tasks. The selected tasks are highlighted.
   - Select a range of tasks by holding the <Shift> key and clicking the first and last tasks in the range. All tasks in between are highlighted.
3. Click **Add** to move the selected tasks from the *Available Tasks* list to the *Activated Tasks* list, or double-click to move the task(s) to the *Activated Tasks* list.
4. Change the order that tasks are listed in the process using two methods:

- Highlight a task, and click the arrow buttons to move the task to a new position on the list.
- Highlight the task and press and hold the mouse button to drag it to another location on the list.

5. To remove one or more tasks, select a task in *Activated Tasks* list using any of the three methods in Step 2 and click **Remove**, or double-click the list to move the task(s) to the *Available Tasks* list.
6. Click **OK**.

The selected tasks are shown in the *Process List* in the Batch Processor interface.

Menu	Tools > Rename Process
Pop-up	Rename Process

7. To give a process a different name, select Rename Process the command.
8. The process name is now ready to edited in the Process List window. Type a new name for the process.

Menu	Tools > Remove Process
Tool	

9. To remove a task from the *Process List*, highlight the task and select the Remove Process command.
10. To move a task to a new position in the *Process List*, highlight the task and press and hold the mouse button to drag it to another location on the list.
11. Select a task to view the options associated with it.

The options associated with the task are displayed in the right side of the interface. You will notice that many of the options resemble the dialog boxes used in the main HIPS interface.

12. Fill in as many task options as necessary (see "BATCH PROCESSOR TASKS" ON PAGE 571).

Process Data

1. Select the Run command.

Menu	Tools > Run
Tool	

When the Run command is selected, you are prompted to save the batch processor file if you have not done so already.

2. Save the Batch Process file.

The results of the Run process are displayed in the Output tab of the Worksheet window.

Batch Processor Tasks

When a task in the Process List is highlighted, the options associated with it are displayed on the right side of the application The fields are often identical to corresponding task in HIPS and SIPS.

Process List tasks

To find more information on any of the following tasks, click a link in the Reference column.

Batch Processor Task	HIPS and SIPS Command	Toolbar Button	Reference
Add to Session	—	—	
Add to Mosaic	Process > New Mosaic		"Add To Mosaic" on page 554
Combine Surfaces	Process > BASE Surface Combine	—	"Combine" on page 429
Compute TPE	Process > Compute TPE		"Compute TPE" on page 255
Apply Filters to Lines	Tools > Set Filters		"TPE Filtering" on page 256 or "Swath Filtering" on page 249 or Single Beam Filtering on page 262
Conversion Wizard	Tools > Conversion Wizard		"File Conversion" on page 169
Finalize Surface	Process > BASE Surface > Finalize	—	"Finalize BASE Surface" on page 427
Load Delta Draft	Process > Load Delta Draft	—	"Delta Draft" on page 268
Compute GPS Tide	Process > Compute GPS Tide	—	"GPS Tide" on page 269
Load Tide	Process > Load Tide		"Tide" on page 227
Load TrueHeave	Process > Load TrueHeave		"TrueHeave Data" on page 237
Merge	Process > Merge		"Merge" on page 272

Batch Processor Task	HIPS and SIPS Command	Toolbar Button	Reference
Modify Queue	—	—	“MODIFY QUEUE” ON PAGE C-22
Recompute BASE Surface	Process > BASE Surface > Recompute		“RECOMPUTE” ON PAGE 423
Recompute Contact Position	Process > Recompute Contact Position	—	“RECOMPUTE CONTACT POSITION” ON PAGE 544
Recompute Towfish Navigation	Process > Recompute Towfish Navigation	—	“RECOMPUTE TOWFISH NAVIGATION” ON PAGE 519
SIPS Template Wizard	Process >SIPS Template Wizard		“SIPS TEMPLATE WIZARD” ON PAGE 557
Sound Velocity Correction	Process > Sound Velocity Correction		“SOUND VELOCITY PROCESSING” ON PAGE 241
Surface Cleaning	Tools > Surface Cleaning		“SURFACE CLEANING” ON PAGE 459

24

Create Publications

Overview

HIPS data can be printed either to paper or Portable Document Format (PDF) files using templates.

Basically, templates are blank forms where data can be positioned using object tags that act as place holders for that data. HIPS contains a group of predefined templates (based on paper size and orientation) in the Session folder. Templates have a RPX file extension.

When a template is opened in HIPS (using the Publish command), the object tags in the template are filled with data from the currently open field sheet or session. The object tag must match the object type open in HIPS.

Object tags are contained in a dictionary file (HIPS.dic) located in the System folder.

You can apply resolution and scale settings for image capture, and select border, grid and scale bar styles. You can change the text values in label objects.

View your finished publication in the Publication Viewer, as well as copy, print and export it.

Publication utilities

Two auxiliary applications for creating and editing templates are included with this installation:

- The **CARIS Publication Designer** is used for building or modifying templates. Dictionary files can be opened in the Publication Designer and tags from those files can be inserted into the template. Other elements such as images or text can also be added for visual effect. The Publication Designer is opened from Start > Programs > CARIS > Utilities > Publication Designer.
- The **CARIS Template Editor** is used for creating or modifying borders, projected grids, and scale bars that can be opened with the template. The Template Editor is opened from Start > Programs > CARIS > Utilities >Template Editor. It can also be opened from HIPS using Edit > Template Editor.

Both the Publication Designer and Template Editor include reference files and help to assist you with these applications.

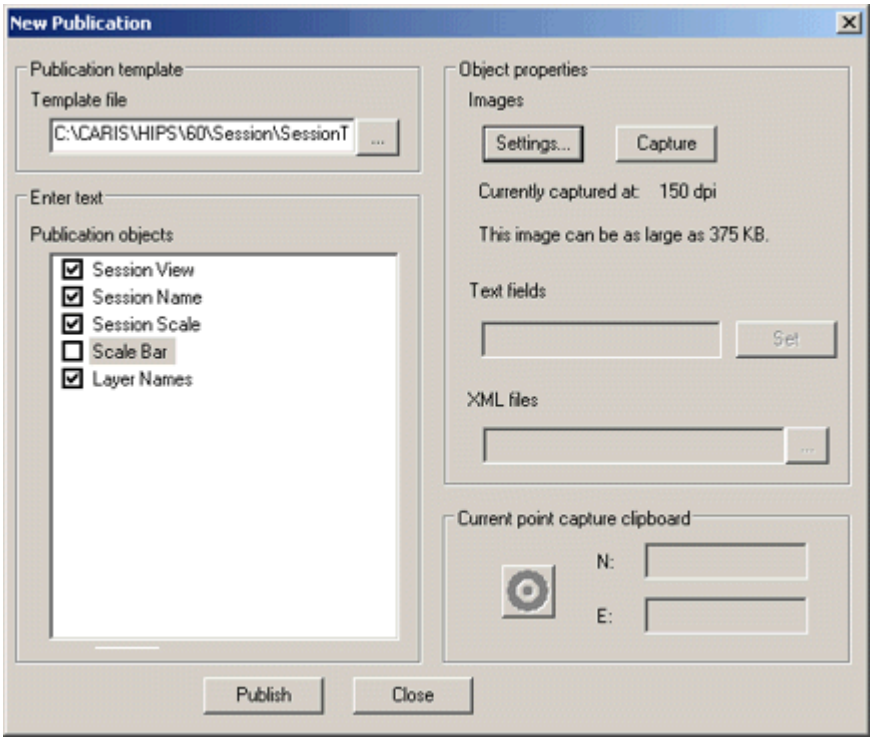
Open a Template

A template contains object tags that act as place holders for data. When a publication is created in HIPS and SIPS, these tags are automatically filled with data from the open session.

Menu	View > Publish
Tool	

1. Make sure that a session file or field sheet is open.
2. Select the Publication Designer command.

The New Publication dialog box is displayed.



3. Click **Browse** to select a template file. The file name and path are displayed in the *Template File* field.

The *Publication Objects* list contains object tags from the HIPS.dic file that were inserted into the template. If an object tag contains a check mark, then the settings are saved in memory and are ready to be opened in the document. Tags representing image objects must always be set before the template is ready to be opened.

To apply the settings for an image object, see "APPLY IMAGE SETTINGS" ON PAGE 577.

To apply the settings for a scale bar, see "APPLY SCALE BAR SETTINGS" ON PAGE 578.

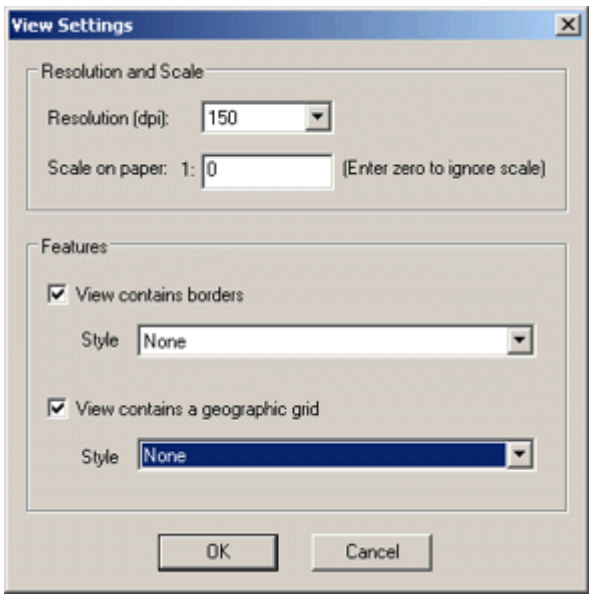
To enter new values for text fields go to "ENTER TEXT VALUES" ON PAGE 579.

4. Otherwise, click **Publish** to view the finished publication in the Publication Viewer.

Apply Image Settings

5. Select an unchecked image object tag so it is highlighted.
6. Click **Settings**.

The Image Settings dialog box is displayed.



7. Select the *Resolution* (DPI) setting for image capture.
8. Select a scale.

The following steps are for selecting the border and grid features.

> **Remember**: You can use the CARIS Template Editor to modify existing borders or grids, or, create custom borders or grids.

9. Select the *View contains borders* check box to select a border.
10. Choose a border from the *Style* drop-down list.
11. Select the *View contains geographic grid* check box to select a grid.
12. Choose a border from the *Style* drop-down list.
13. Click **OK** to save the settings and close the dialog box.

The **Capture** button in the New Publication dialog box is now active.

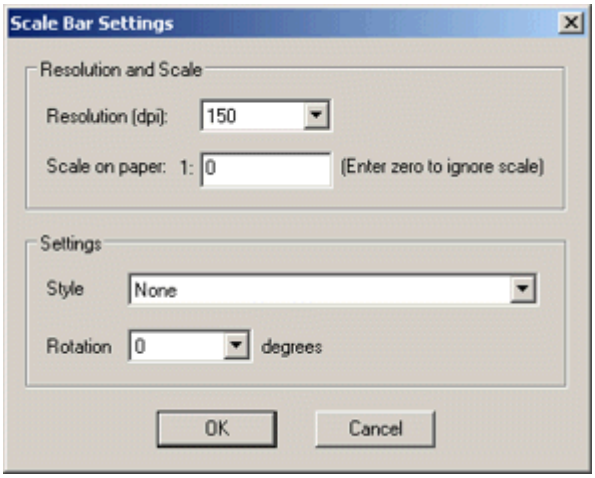
14. Click **Capture** to commit the above settings to memory.

The image object now has a check mark beside it.

Apply Scale Bar settings

15. Select an unchecked scale bar tag.
16. Click **Settings**.

The Scale Bar Settings dialog box is displayed.



17. Select a resolution for the scale bar.
18. Type a scale.
19. Choose a scale bar type from the Style drop-down list.

> **Remember**: You can use the CARIS Template Editor to modify existing scale bar styles, or, create new scale bar styles.

20. Select a degree value from the drop-down list. to rotate the scale bar
21. Click **OK**.

The **Capture** button in the New Publication dialog box is now active.

22. Click **Capture** to commit the above settings to memory.

The scale bar object now has a check mark beside it.

Enter text values

23. If you want to change the value in a label object tag, select the object in the Publication Objects list so that the *Text Fields* field is active.
24. Type a new value in the field and click **Set**.

The value for that object is now changed.

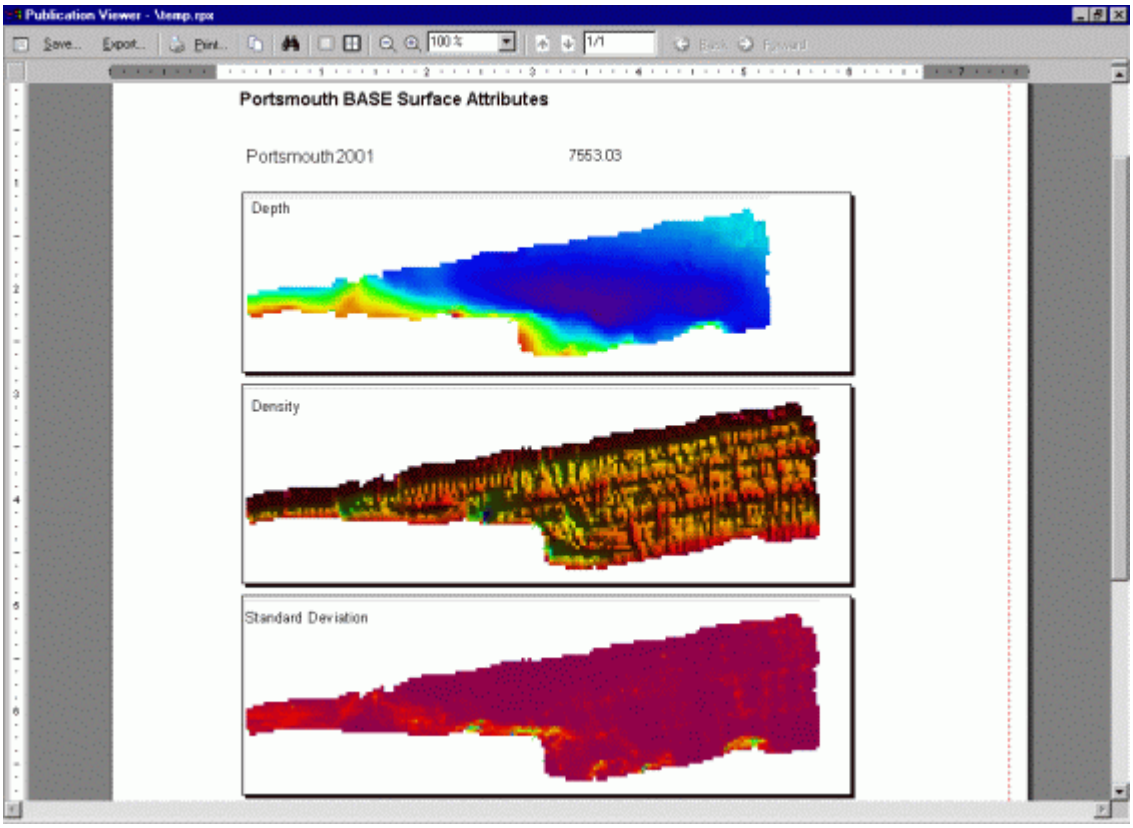
(Note that the *XML files* and *Current point capture clipboard* options are not available in this implementation of HIPS and SIPS.)

25. Click **Publish** to view the finished publication.

The Publication Viewer window is opened.

Publication Viewer

The HIPS and SIPS data is displayed in the Publication Viewer.

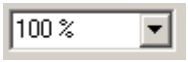
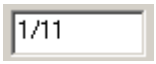


Publication Viewer commands

The following table contains an outline of the commands used in Publication Viewer.

Command	Tool	Purpose and Procedure
Save	—	Save a template file under a new name or to another location. 1. Select the Save command. The Save dialog box is displayed. 2. Select a directory where you want to save the template file. 3. Type a name for the file. 4. Click **Save**. The file is saved.

Command	Tool	Purpose and Procedure
Export	—	Export the template to another format. 1. Select the Export command. The Export dialog box is displayed. 2. To select a new file path, click **Browse** and select a network path in the *Save in* field. 3. In the *File Name* field, type a name. In the *Save as type* field, select one of the following options: • Adobe Acrobat (*.pdf) • Rich Text Format (*.rtf) • Microsoft Excel (*.xls) • Hypertext (*.htm) • Text file (*.txt) • TIFF image (*.tif) 4. Click **Save**. The template is exported to specified location, where you can open and view it using any software that supports the file type you specified.
Print		The Print command displays the Print dialog box, from which you can print the template to a printer, plotter or file location. 1. Select the Print command. The Print dialog box is displayed. 2. Perform one of the following steps: • To print to a network printer, select a printer in the Select Printer. • To print to a network storage location, select the Print to file option. 3. In the Page Range section, select one of the following options: • All • Selection • Current Page • Pages 4. Click **Print** to print the template to the specified location.
Copy		Send a copy of the template to the operating system's clipboard. From the clipboard, you can paste the template in any application that supports the template's formatting. 1. Select the Copy command. A copy of the template is placed on the clipboard.
Find		You can search the template for text that matches specified search criteria. The Find command displays a dialog box in which you can specify search criteria. 1. Select the Find command. The Find dialog box appears. 2. In the Find what field, type the text string to use as search criteria. 3. Optional: Select one of the following search refinement tools: • Match whole word only • Match case • Direction: • Up • Down 4. Click **Find Next**. The Find feature searches the template for text that matches the specified search criteria. When a text string is encountered that matches the search criteria, it is highlighted. 5. Perform one of the following steps: • Click **Find Next** to search for the next instance that matches the specified search criteria. • Click **Cancel** to close the Find dialog box.

Command	Tool	Purpose and Procedure
Single Page		The Single Page tool displays one template page per display screen. 1. To display the template in single page mode, select the Single Page command.
Multiple Pages		The Multiple Page tool displays multiple template pages per screen. 1. To display the template in multiple page mode, select the Multiple Page command. The display refreshes to display multiple pages at a time.
Zoom Out		Zoom out of the template display. 1. To zoom out of the template display, select the Zoom Out command.
Zoom In		Zoom into the template display. 1. To zoom into the template display, select the Zoom In command. The display refreshes to display the template at a larger scale.
Custom Zoom		Zoom into or out of the template display using a preset factor. To zoom into or out of the display using a preset factor, do one the following steps: • Type a percentage value in the custom zoom field; • select one of the preset Zoom factors in the drop down list. The display refreshes to re-display the template at its new zoom setting.
Previous Page		The Previous Page tool reverts the display to the previous template page. 1. Select the Previous Page command. The display reverts to the previous template page.
Next Page		The Next Page tool advances the display to the next template page. 1. Select the Next Page command. The display advances to the next template page.
Custom Page		Advance the display to a specified page. 1. Type a new page value in the custom page field. The display refreshes to show the specified template page.
Move Backward		Using the Move Backward command, you can navigate through the template pages in non-sequential order. For example, if the previously displayed template page is page 5 of 11 (5/11) and you are currently viewing page 10 of 11 (10/11), the Move Backward command displays page 5 of 11. 1. Select the Move Backward command. The previously displayed template page is displayed.
Move Forward		The Move Forward tool reverts to the page that was displayed before the Move Backward command was used. 2. Select the Move Forward command. The page that was displayed before the Move Forward command was used is displayed again.

Template Editor

The Template Editor defines borders, scale bars, and projected grids that can be saved and applied when creating templates for publication.

Template Editor files can be saved using any name but must have the following file extensions.

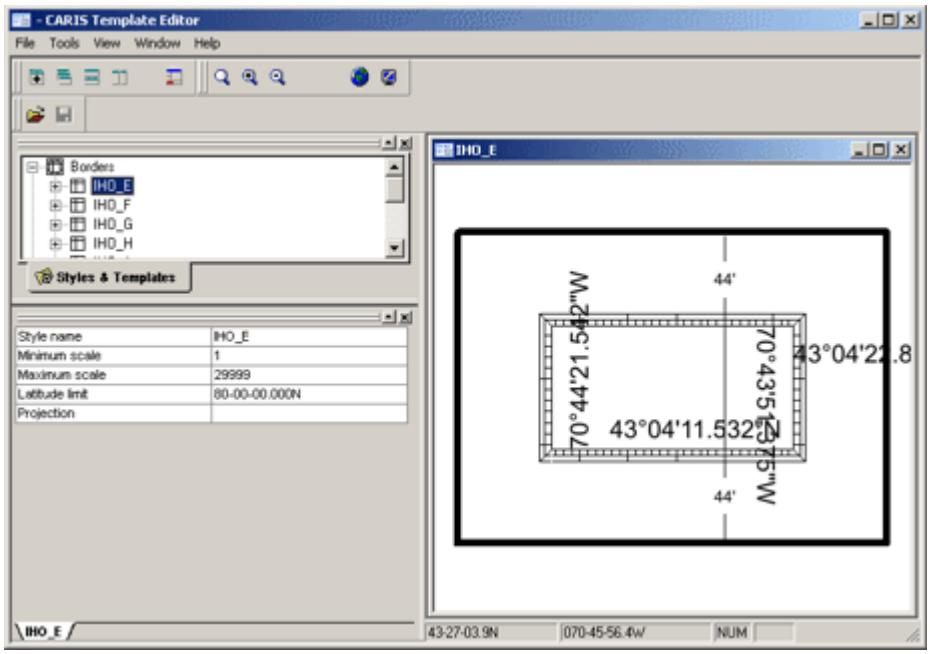
Template Component	File Extension
Borders	.bsd
Scale Bar	.ssd
Projected Grid	.grd

Default templates are stored in the ..\System directory where HIPS is installed. Your customized template files can be stored in any directory you choose

1. Select the command.

Menu	Edit > Template Editor

The Template Editor is displayed.



For more information, see the Template Editor help that is available in that application.

25

Export Data

Introduction

The Export Wizard converts HIPS data products (soundings, BASE Surfaces, mosaics, etc.) to the following formats:

- ASCII text file
- CARIS map
- Bathymetric Attributed Grid (BAG)
- Generic Sensor Format (GSF) files
- Hydrographic Object Binary (HOB) file
- Hydrographic Transfer Format (HTF) files
- TIFF image
- Unified Sonar Imaging Processing System (UNISIPS) format

The HIPS Export Wizard is a separate application launched from the HIPS and SIPS interface.

The number of dialog boxes displayed by the Export Wizard depends on the format selected in the first dialog box.

t

Menu	File > Export
Tool	

1. Select the t command.

The Export Wizard is displayed.

See the table below to view export options.

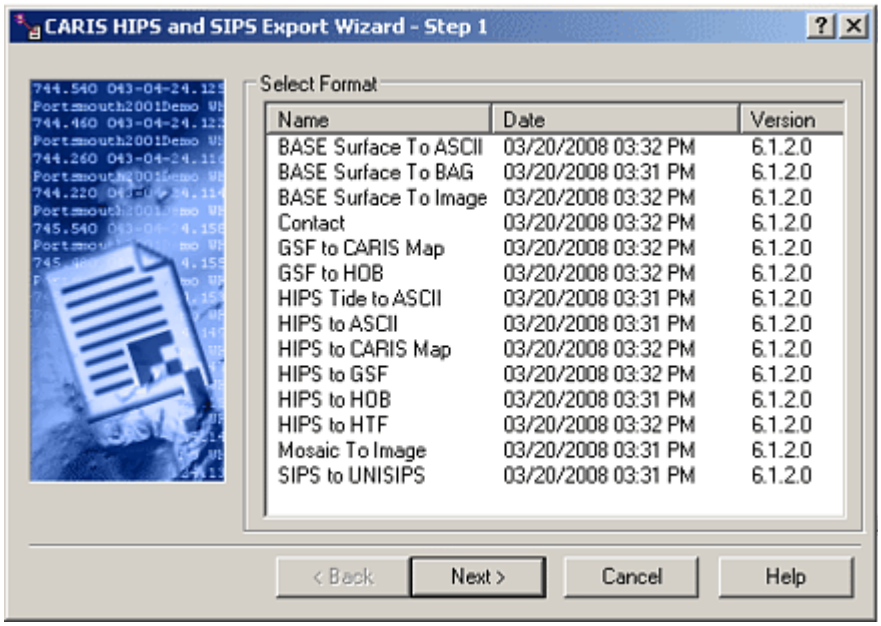
Formats

Format	Description	Refer to...
BASE Surface to ASCII	Export BASE Surface position and attribute data to a text file.	"BASE Surface To ASCII" on page 591
BASE Surface to BAG	Export BASE Surface elevation and uncertainty values to Bathymetric Attributed Grid (BAG) file	"BASE Surface to BAG" on page 593
BASE Surface to Image	Export BASE Surface attribute data as a georeferenced image for further processing in a CARIS product or other application.	"BASE Surface To Image" on page 595
Contacts	Export contacts to a CARIS map, an ASCII file, and/or a TIFF image.	"Contacts" on page 597
GSF to CARIS	Export soundings and tracklines from one or more Generic Sensor Format (GSF) files to a CARIS map. The GSF data does not have to be converted to HIPS format prior to using this utility.	"GSF to CARIS Map/HOB" on page 600

Format	Description	Refer to...
GSF to HOB	Export GSF data to a Hydrographic Object Binary (HOB) file that stores feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects.	"GSF TO CARIS MAP/HOB" ON PAGE 600
HIPS Tide to ASCII	Export tide data from track lines to text format.	"HIPS TIDE TO ASCII" ON PAGE 604
HIPS to ASCII	Export HIPS data to a customized text listing of soundings that you can load into other software systems.	"HIPS TO ASCII" ON PAGE 605
HIPS to CARIS	Export HIPS data to a CARIS map. This option contains three modes of exporting data: soundings, tracklines and swaths.	"HIPS TO CARIS MAP" ON PAGE 608
HIPS to GSF	Export processed HIPS data to GSF format. A new GSF file is created for each trackline that is exported from HIPS. Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function updates these GSF files.	"HIPS TO GSF" ON PAGE 612
HIPS to HOB	Export HIPS data to HOB files that store feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects. This option contains three modes of exporting data: soundings, tracklines and swaths.	"HIPS TO HOB" ON PAGE 613
HIPS to HTF	Export data to a Hydrographic Transfer Format (HTF) file. The HTF is a text file that contains a header section and a sounding record. The Export wizard can save a header section so it can be loaded into multiple HTF files.	"HIPS TO HTF" ON PAGE 615
SIPS to UNISIPS	Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.	"SIPS TO UNISIPS" ON PAGE 618
Mosaic to Image	Export a mosaic as a TIFF image for further processing in a CARIS product or other application.	"MOSAIC TO IMAGE" ON PAGE 619

Select Format (Step 1)

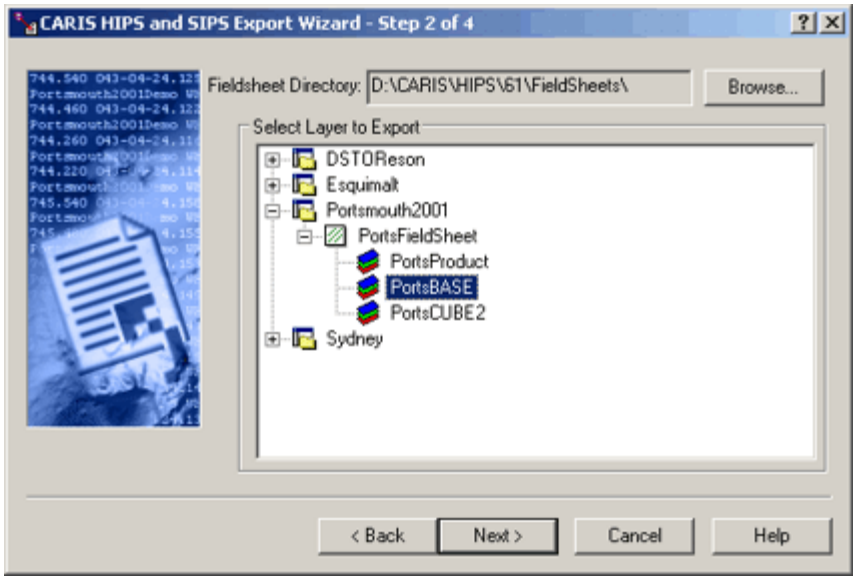
The Select Formats dialog box displays the available formats for exporting data.


CARIS HIPS and SIPS Export Wizard - Step 1
Select Format
Name
Date
Version
BASE Surface To ASCII 03/20/2008 03:32 PM 6.1.2.0
BASE Surface To BAG 03/20/2008 03:31 PM 6.1.2.0
BASE Surface To Image 03/20/2008 03:32 PM 6.1.2.0
Contact 03/20/2008 03:32 PM 6.1.2.0
GSF to CARIS Map 03/20/2008 03:32 PM 6.1.2.0
GSF to HOB 03/20/2008 03:32 PM 6.1.2.0
HIPS Tide to ASCII 03/20/2008 03:31 PM 6.1.2.0
HIPS to ASCII 03/20/2008 03:31 PM 6.1.2.0
HIPS to CARIS Map 03/20/2008 03:32 PM 6.1.2.0
HIPS to GSF 03/20/2008 03:32 PM 6.1.2.0
HIPS to HOB 03/20/2008 03:31 PM 6.1.2.0
HIPS to HTF 03/20/2008 03:32 PM 6.1.2.0
Mosaic To Image 03/20/2008 03:31 PM 6.1.2.0
SIPS to UNISIPS 03/20/2008 03:31 PM 6.1.2.0
< Back
Next >
Cancel
Help

1. Select an export format.
2. Click **Next**.

Select Files (Step 2)

The second dialog box lets you select the data for export.



The options in the dialog box and the data that it displays vary with the product you are exporting, and the export choice you made at Step 1.

BASE Surface/Mosaic/Grid/BAG

The dialog box displays all project files with associated field sheets and layers (grids, contacts, mosaics, etc.).

1. Expand a project file tree and select the layer you want to export (contact, grid, mosaic, or BASE Surface). If you are exporting a mosaic to an image, you can select more than one mosaic for export by holding the Ctrl button while selecting mosaics.

Contacts/HIPS soundings/Tide/SIPS

The dialog box displays the existing projects. All HIPS and SIPS projects are organized in a Project/Vessel/Day/Line structure.

1. Select all or some of the files in the Project/Vessel/Day/Line directory. Click the expand icon (+) icon to view the file tree, if needed.

HIPS to HTF

The dialog box displays either a list of projects and associated field sheets or the Project/Vessel/Day/Line data structure depending on whether you select the *Export Field Sheet Product* or *Export Raw Data* options.

1. Select the *Export Field Sheet Product* option to export layer data or select the *Export Raw Data* option to export sounding data from tracklines.

If you selected the *Export Raw Data* option, the *Include Rejected* option is active. This option includes rejected soundings in the export process.

2. [Optional] Select the *Include Rejected* option to export rejected data.

If the *Export Raw Data* option is selected, the file tree is refreshed to show the Project/Vessel/Day/Line directory structure.

3. Open a project or session file tree and select a layer or trackline data.

GSF

The dialog box displays a section for the file path and name of the GSF file

1. Click Select to choose the GSF files to be imported to CARIS/HOB format.

The file path and name is displayed in the dialog box.

2. To remove a file, select it so it is highlighted and click Remove.

The file is no longer visible.

BASE Surface To ASCII

Export BASE Surface attributes to an ASCII text file.

If you exporting attribute data from a BASE Surface or weighted grid created prior to HIPS and SIPS 6.1, the Step 4 dialog box is not displayed because the default measurement unit is metres.

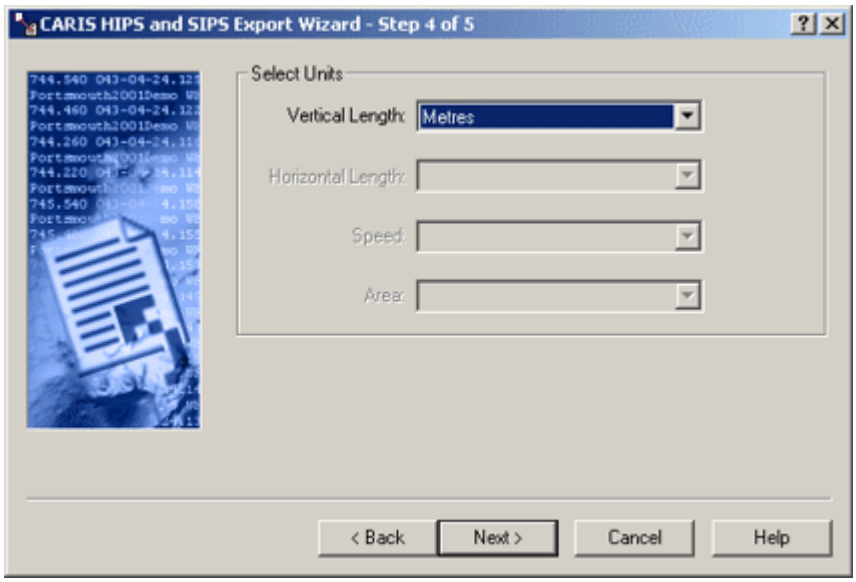
BASE Surface to ASCII (Step 3)

1. Type a path and name for the file, or click **Browse** to select a folder and then type the file name.
2. Select the position units for data from the drop-down list:
   - ground units (east, north, depth)
   - geographic DMS (latitude, longitude, depth)
   - geographic DMS (no formatting)
   - Geographic DD (decimal degrees)
3. Select how many digits are displayed to the right of the decimal point by selecting a number using the *Precision* up or down arrow buttons.
4. Select the BASE Surface attribute values to be recorded in the ASCII file by selecting the attribute in the *Available* column and clicking **Add** to move it to the *Active* column.
5. To remove an attribute, select the attribute from the *Active* column and click **Remove** to move it back to the *Available* column.
6. Choose a *Attribute Precision* value (number of digits to the right of decimal point) for attributes in the *Active* column by selecting an attribute and clicking the up or down arrow buttons.

7. Select the *Headers* check box to create print attribute names at the top of the attribute rows.
8. Select either the *Space*, *Comma*, or *Tab* check boxes to use these characters as delimiters, or select the *Other* check box and type a character to use a delimiter.
9. Click **Next**.

BASE Surface To ASCII (Step 4)

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.

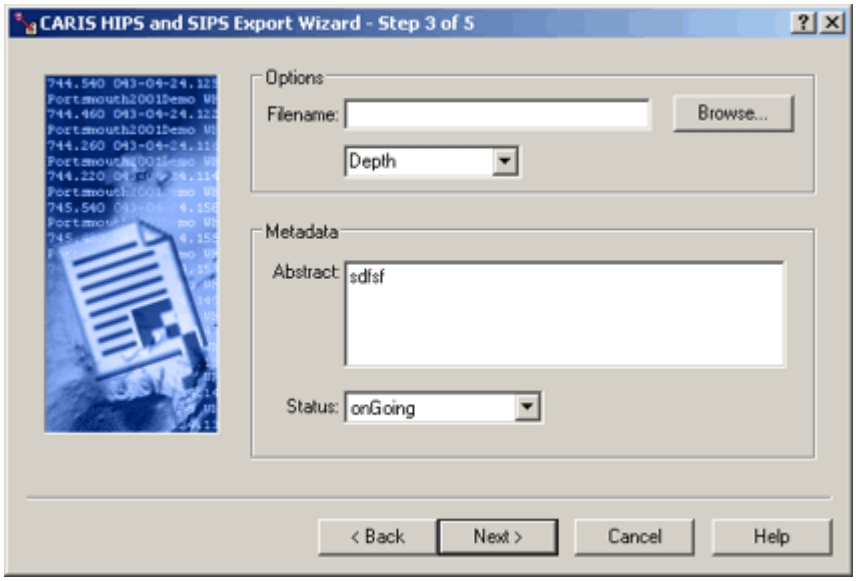


1. Select the units for recording BASE Surface data:
   - metres and kilometres
   - fathoms (US and international)
   - feet (US and international)
   - yards (US and international)
   - miles (US, international, and nautical)
2. Click **Next**.

BASE Surface to BAG

Export a BASE Surface to a Bathymetric Attributed Grid file.

BASE Surface to BAG (Step 3)



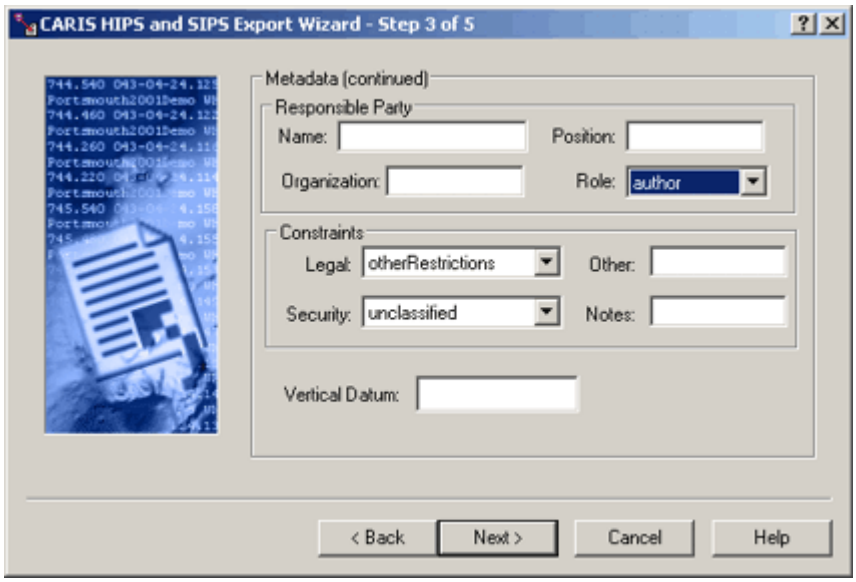
1. Type the path and name for the BAG file, or click **Browse** and select a directory and name.
2. Select the BASE Surface attribute values to export from the drop-down list.

Metadata is general information about the file (the who, what, where, why, and how).

3. Type any information in the *Abstract* field.
4. Select a *Status* from the drop-down menu.
5. Click **Next**.

BASE Surface to BAG (Step 4)

This dialog box is a continuation of the metadata section.

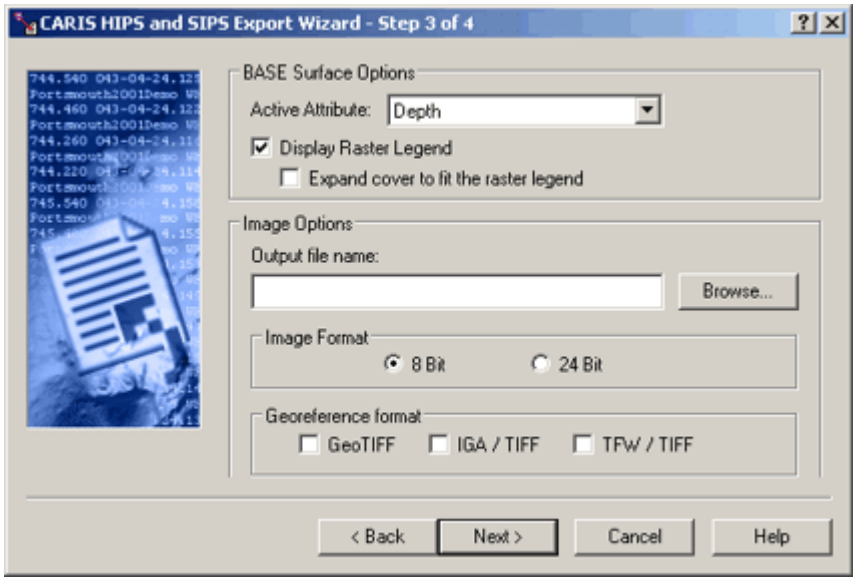


1. Type personal information in the *Name*, *Position*, and *Organization* fields; select a *Role* from the drop-down menu.
2. Select a *Legal* constraint from the drop-down list. If the otherRestrictions option is selected, you can type it into the Other field.
3. Select a *Security* constraint from the drop-down list. Type other information in the *Notes* field.
4. Type a *Vertical Datum*. Remember that the values above the vertical datum are positive.
5. Click **Next**.

BASE Surface To Image

Select the following links to export a BASE Surface to an image file.

BASE Surface to Image (Step 3)



1. Select a BASE Surface attribute to display as a georeferenced image.

If the selected attribute has a legend associated with it, you can include this legend in the image. The legend will be drawn according to the property settings. If the cover of the field sheet matches the cover of the BASE Surface, the field sheet cover can be expanded to accommodate the legend so it does not cover the surface.

2. Select the *Display Raster Legend* check box to draw a raster legend for the selected attribute.
3. Select the *Expand cover to fit the raster legend* check box to expand the area cover so that the legend does not overlay the surface.
4. Click **Browse** to select the file path and name of the image file.
5. Select a colour format for the image.
   - *24 Bit (*16 million colours)
   - *8 Bit* (256 colours)

The Surface can be saved as a TIFF file with either an IGA or a TFW georeferenced file, or both. You can save the image as a GeoTIFF image that records the georeferenced information directly in the .TIFF file.

6. Select a geo-reference format for the image:
   - *GeoTIFF*
   - *IGA/TIFF*
   - *TFW/TIFF*
7. Click **Next**.

Contacts

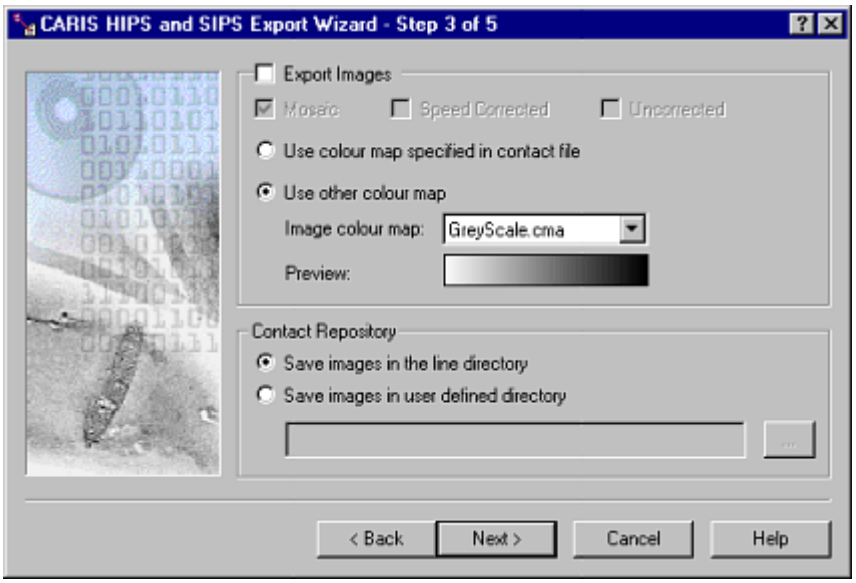
Export contacts to a CARIS map and/or an ASCII text file. You can also export contact snapshot images to TIFF files. The ASCII files are structured to be easily imported into a relational database.

If you select ASCII, you will have to browse to a folder location and type a file name. This name is used as the first component name of four files that will be created to store the contact information:

- **filename_ContactLine.txt** contains information identifying the survey line and indexes to the other files.
- **filename_ContactSingle.txt** contains all the information for single point contacts.
- **filename_ContactMulti.txt** contains all information for single point contacts.
- **filename_ContactMultiPoints.txt** contains all location information for multipoint contacts.

For more information on the structure of contact files, see "CONTACT FILE FORMATS" ON PAGE 657.

Contacts (Step 3)



1. Select the *Export Image* check box to export the contact(s) in TIFF image format.

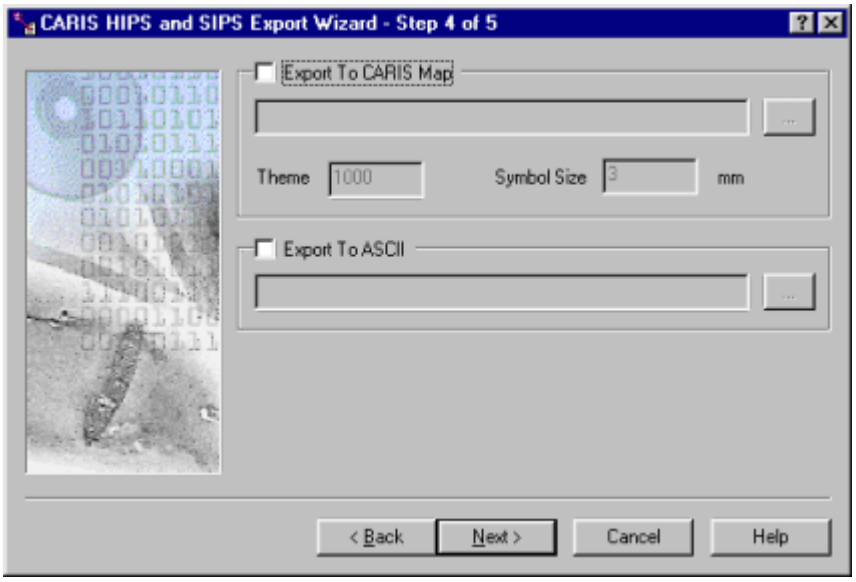
2. If you select *Export Image*, select from the following image formats:
   - *Mosaic*: This option creates a georeferenced TIFF image from contact data.
   - *Speed Corrected*: This option calculates the distance between profile lines in the contact and corrects for vessel speed.
   - *Uncorrected*: This option does not correct for vessel speed and renders a one-to-one correspondence between the side scan image and the contact.
3. To export the colour map that was selected in the Side Scan Editor, select the *Use Colour Map Specified in Contact File*.
4. To export the contact with a new colour map, select the *Use Other Colour Map* option and select the map from the drop-down list.

The selected colour map is displayed in the *Preview* field.

5. Select the directory location for the contact. You can select from two options:
   - *Save Images in the Line Directory*: This option saves the contact data to the line file in the HIPS/SIPS Project/Vessel/Day directory structure that you selected in the second dialog box.
   - *Save Images in User Defined Directory*: This option saves the contact data in another directory. Click **Browse** to select the folder where you want to save the data.
6. Click **Next**.

Contacts (Step 4)

This dialog box contains options for exporting contact data to a HOB or CARIS file

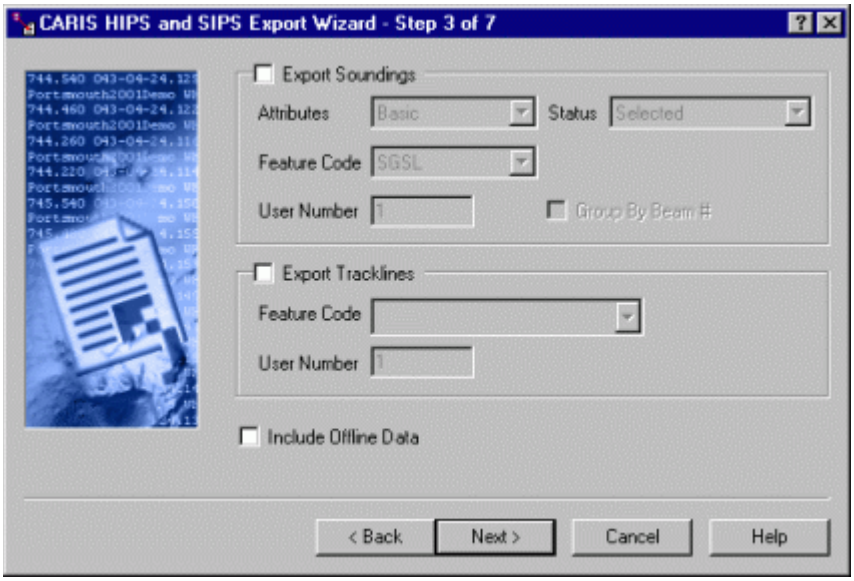


1. Select either the *Export to CARIS Map* or *Export to ASCII* check boxes. Both can be selected at the same time.
2. Type a file name and path to store the contact data after export, or click **Browse** to select the file path (both CARIS data and ASCII text files are typically stored in ..\Hips\CARIS_Files).
3. Type a layer number for the contact data in the *Theme Number* field.
4. Type a symbol size for the contact data in the *Symbol Size* field.
5. Click **Next**.

GSF to CARIS Map/HOB

The Generic Sensor Format (GSF) to CARIS/HOB function loads soundings and track lines from one or more GSF files into a CARIS map or HOB file. GSF data does not have to be converted to HIPS data prior to using this utility.

GSF to CARIS/HOB (Step 3)



Data can be exported either in the form of soundings or track lines:

- **Soundings**. Export all soundings (subject to the HIPS Sounding Status and Data Thinning settings) that fit within the boundaries of the work file.
- **Track Lines**. Export a line that connects the centre of each swath for each survey line, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

1. Select either the *Export Soundings* or the *Export track lines* check box.
2. If you selected *Export Soundings*, complete any or all of the following fields:
   - *Attributes*: Select *Basic* or *Advanced* from the drop-down list.

     The *Basic Attributes* option creates mandatory CARIS feature attributes for the map file, such as feature code, user number and source ID.

The *Advanced Attributes* option creates additional CARIS feature attributes for soundings such as time stamps, launch identifier and tidal correction.

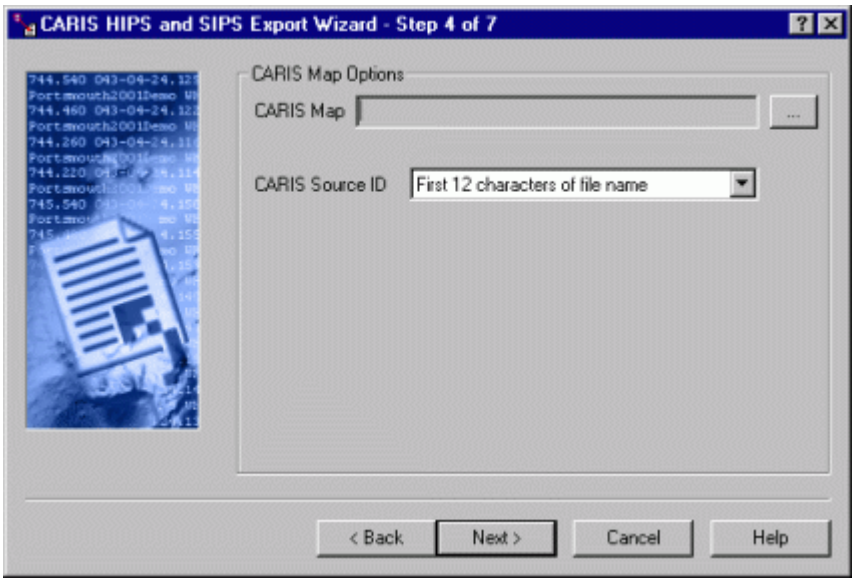
- *Status*: Select soundings flagged as Selected, Accepted or Accepted and Rejected.
- *Feature Code*: Select a feature code for the soundings from the drop-down list.
- *User Number*: Type a number value for layering data in the CARIS map or HOB file.
- *Group By Beam*: Load soundings into user numbers according to sonar beam number. For example, all soundings from beam 37 are loaded to user number 37 in the CARIS map file or HOB file

3. If you selected *Export Tracklines*, complete any or all of the following fields.
   - *Feature Code*: Select a feature code for the track lines object from the drop-down list.
   - *User Number*: Type a number value for layering data in the CARIS map.
4. Select the *Include Offline Data* check box to include data recorded during turns between track lines.
5. Click **Next**.

The next dialog box is determined by whether you are exporting to a CARIS map or HOB file.

GSF to CARIS (Step 4)

This dialog box is displayed if you are exporting to a CARIS map.



1. Click **Browse** to select a CARIS map for data export, or type the name of a new map.

The map file path and name are displayed in the *CARIS Map* field.

The *CARIS Source ID* is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a GSF project name.

2. Choose either the first 12 characters or the last 12 characters for the *CARIS Source ID*.
3. Click **Next**.

If you created a new CARIS map in Step 4, a projection list is displayed in the next dialog box. Otherwise you can begin export of the data in the next dialog box.

GSF to HOB (Step 4)

This dialog box is for saving the HOB file that contains the exported GSF data.

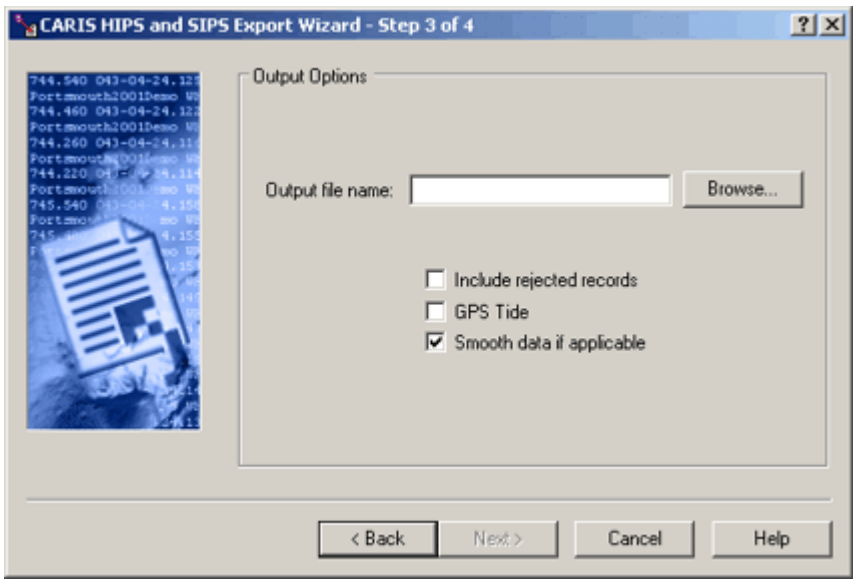
1. Click **Browse** and select a location and name for the HOB file.
2. Click **Next**.

HIPS Tide to ASCII

Export HIPS tide data from selected track lines to a TID file. You can create TID files from single or multiple track lines in a project. TID files can be viewed in a text editor.

HIPS Tide to ASCII (Step 3)

This dialog box creates the output tide file.



1. Click **Browse** and use the standard Windows Open dialog box to select a path and name for the output TID file.

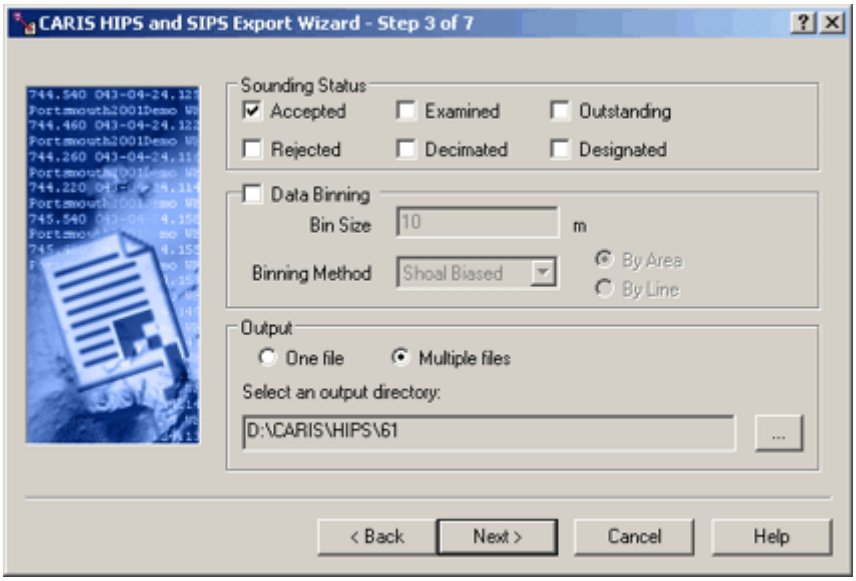
The path and name of the tide file is displayed in the *Output File Name* field.

2. Select the *Include Rejected Records* check box to include tide records that have been rejected.
3. Select *GPS Tide* to use tides calculated from a GPS ellipsoid height.
4. Select *Smooth data* to use tide data that has been smoothed for localized variability.

HIPS To ASCII

The HIPS to ASCII function creates a customized text listing of soundings that you can load into other software systems.

HIPS to ASCII (Step 3)



Each sounding in a cleaned HIPS dataset are stored with bits that determine the status of that sounding. You can choose to restrict the export process to soundings with specific status flags.

1. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.
2. Choose the sounding's unit of measurement by selecting either *Metres* or *Feet*.

The *Data Binning* option divides the survey area into a grid to select the shoal- or deep-biased soundings from each cell for export. The *Bin Size* option sets the size of the grid cells.

3. To apply a data thinning function, select the *Data Binning* check box.
4. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.
5. Select either the *Shoal-Biased* or *Deep-Biased* options to export only the shoalest or deepest soundings, respectively, to the ASCII file.

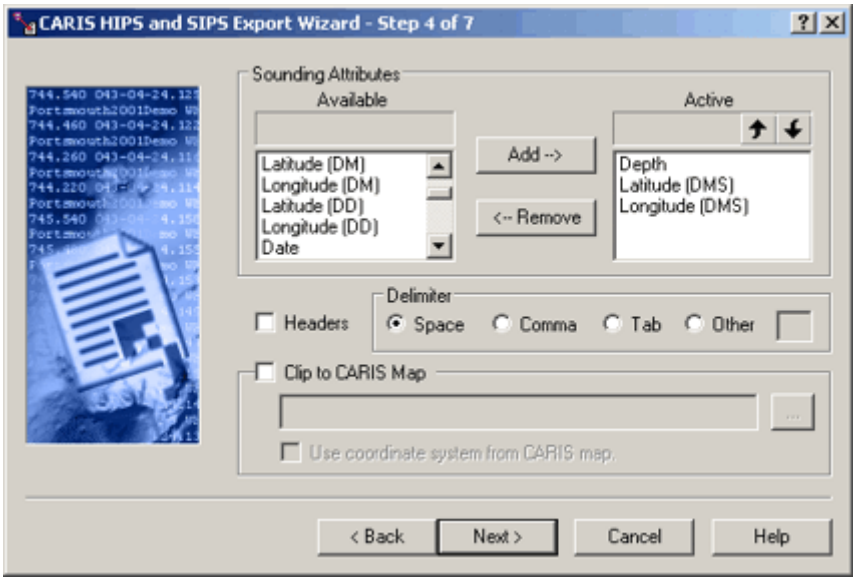
Soundings can be selected for export from each grid cell in the entire survey area or by processing each track line in sequence.

6. Select either the *By Area* or *By Line* options.

7. Select *One File* to export all data to a single file or select *Multiple Files* to export data to separate files for each survey line.
8. Select an output location for the data.
   - If you select *One File*, type a name and file path for the ASCII file, or click **Browse** to select a location and name for the file.
   - If you select *Multiple Files*, click **Browse** to select a folder to save the files. The files are automatically named according to Project_Vessel_Date_SurveyLine.
9. Click **Next**.

HIPS to ASCII (Step 4)

This dialog box lists the attributes that can exported to the ASCII file and the delimiters used to separate data in the file.



1. Select the sounding attributes to be included in the ASCII file.
   - To add an attribute, select the attribute in the *Available* list and click **Add** to move it to the *Active* list.
   - To remove an attribute, select the attribute in the *Active* list and click **Remove** to move it to the *Available* list.
2. To change the order that attributes are exported, select the attribute in the *Active* list and click the up or down arrow buttons.
3. Select the *Headers* check box to create print attribute names at the top of the attribute rows.
4. Select either the *Space*, *Comma*, or *Tab* check boxes to use these characters as delimiters, or select the *Other* check box and type a character to use as a delimiter.

You can define the boundaries of the ASCII map data from the boundaries of an existing CARIS map. If you do not select this option, the extents are defined from the ASCII data.

5. Select the *Clip to CARIS Map* check box to use a CARIS map to define the boundaries of the ASCII map data.
6. Click **Browse** to select an existing CARIS map.
7. To use the coordinate system from the selected CARIS map for the ASCII data, click the check box.

If you selected the *Use Coordinate System From CARIS Map* option, the next dialog box prompts you to export the data. If you did not select this option, the next dialog box prompts you to select a coordinate system.

8. Click **Next**.

HIPS To ASCII (Step 5)

This dialog box sets the unit type for recording data. The fields that are active in this dialog are determined by the attributes selected in the previous dialog box.

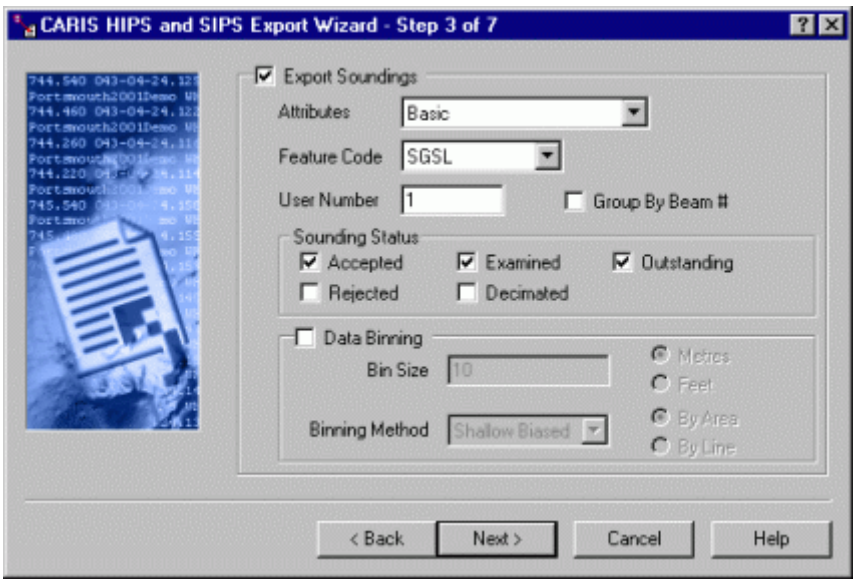
1. Select the units for recording sounding data:
   - metres and kilometres
   - fathoms (US and international)
   - feet (US and international)
   - yards (US and international)
   - miles (US, international, and nautical)
2. Click **Next**.

HIPS To CARIS Map

Export data to a CARIS map.

HIPS to CARIS (Step 3)

You can select which soundings are to be exported and how they will be stored in the CARIS map.



The *Export Soundings* option exports all soundings (subject to the sounding status and data binning settings).

1. Select the *Export Soundings* check box if you want to export soundings to a CARIS map (see the next dialog box for the *Export Tracklines* and *Export Swaths* options).

There are four methods for exporting sounding attributes to a CARIS map.

- *Basic*: The basic sounding attributes in the CARIS map. These include user number, feature code, and source ID. This information is adequate to generate products such as sounding plots, contour plots, and DTMs.
- *Extended*. Basic attributes plus time stamps, launch identifier (first four letters of the vessel name), and tidal correction.
- *Extended with Key*: The extended attributes set, plus the profile number and beam number as the CARIS Key. The profile number and the beam number, in addition to the other attributes, allow each sounding to be identified uniquely with respect to the original data set.

- *Extended with Unique Key*: The extended attributes set, plus every object is given a unique key.

This additional information allows more flexibility for visualizing and manipulating CARIS files. For example, if time stamps are associated with each sounding, you can limit the display of soundings based on a time range.

However, each extra attribute adds to the storage requirements. More disk space is required to store the work file, and any HIPS program that processes the file must do more work and use more time and CPU resources.

2. Select from the four options in the *Attributes* drop-down list.
3. Select a new feature code for the soundings, if needed.

The *Group by Beam Number* option layers soundings according to a beam number in the CARIS file. Soundings belonging to a beam are assigned to the same user number in the map. User numbers in CARIS file are similar to layering. This option is used when exporting soundings from checklines into a separate checkline map so that a Quality Control Report can be generated. If you do not select this option, then a single user number will be used to hold all the exported soundings.

4. Select the *Group by Beam Number* check box to layer soundings according to beam numbers in the CARIS map.

The *User Number* option is available only if you do not select the *Group by Beam Number* option.

5. [Optional] Type a *User Number* to which all soundings are to be assigned.
6. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells.

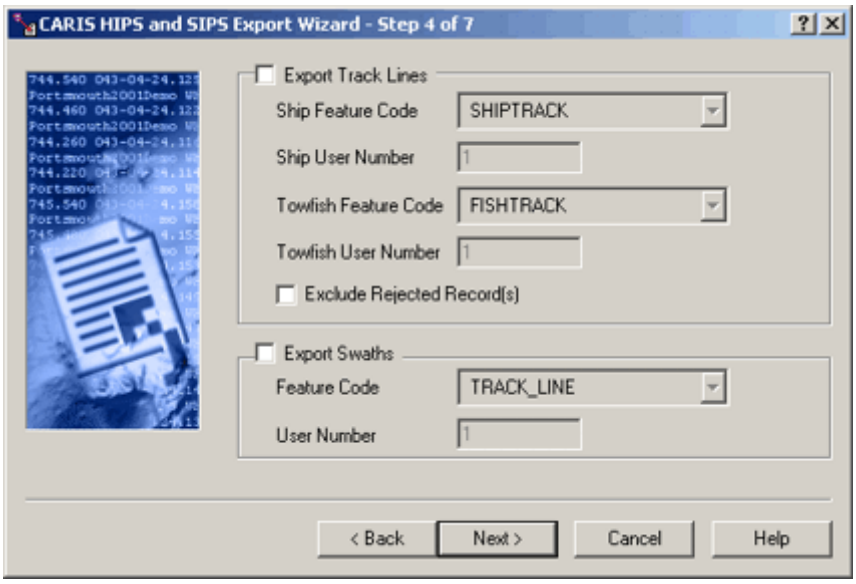
7. To apply a data binning function on export, select the *Data Binning* check box.
8. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.
9. Select either *Shoal-Biased* or *Deep-Biased* to export only the shoalest or deepest soundings from each cell, respectively, to the CARIS file.

Soundings can be selected for export from the total survey area or by processing each track line in sequence.

10. Select either the *By Area* or *By Line* options.
11. Click **Next**.

HIPS to CARIS (Step 4)

This dialog box sets the track line and swath export options.



The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to import soundings to use this option.

The *Export Swaths* command connects a set of lines (one per swath) along the outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to import soundings to use this option.

1. Select the *Export Track Lines* check box to export a line representing the track of the vessel along the survey area.
2. Type a new feature code for the vessel, if needed.
3. Type a *User Number* for the vessel so it is distinguished from other vessels in the same CARIS map.
4. Select a new *Towfish Feature Code*, if needed.
5. Type a *User Number* for the towfish so it is distinguished from other vessels in the same CARIS map.
6. Select the *Exclude Rejected Record(s)* check box to not include this data.
7. Select the *Export Swaths* check box to export a set of lines that represents the swaths along the survey.
8. Type a *User Number* for the swath so it is distinguished from other swaths in the same CARIS map.
9. Click **Next**.

HIPS to CARIS (Step 5)

1. Click **Browse** to select a CARIS map for data export, or type the name of a new map.

The map file path and name are displayed in the *CARIS Map* field.

The Source ID is a 12-character alphanumeric attribute in CARIS maps. The ID can be the first or last 12 characters in a project name or line name.

2. Select either *Project Name* or *Line Name* to use as a Source ID.
3. Choose either the first 12 characters or the last 12 characters for the Source ID.

The *Clip to CARIS Map* option cuts the HIPS data at the boundaries of an existing CARIS map. If this option is not selected, the geographic extents of the map are extended to accommodate the HIPS data.

4. Select the *Clip to CARIS Map* check box to implement this option.
5. Click **Next**.

If you created a new CARIS map in Step 1, a dialog box for selecting a projection is displayed. Otherwise you can begin export of the data in the next dialog box.

HIPS To GSF

The HIPS to GSF function exports processed HIPS data to a GSF format. A new GSF file is created for each track line that is exported from HIPS.

Conversely, when HIPS data is created from GSF data, a copy of the original GSF file is maintained inside the HIPS directory structure. The Export to GSF function updates these GSF files.

HIPS to GSF (Step 3)

1. Click the **Browse** button to select a location for the GSF files.

The Browse for Folder dialog box is displayed. The stored GSF file will have the same name as the track line.

2. Select a directory to store the GSF data and click **OK** to close the dialog box.

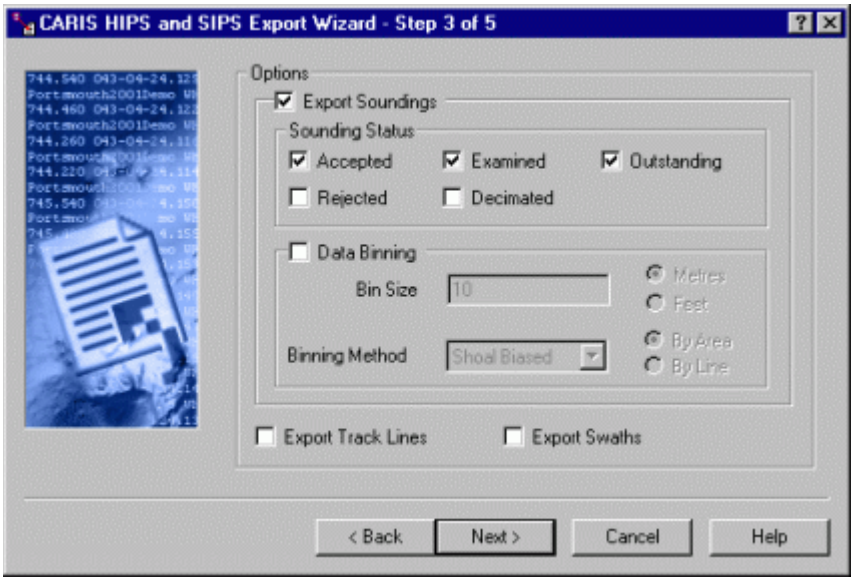
The file path is displayed in the *Output File* field.

3. To export entire projects and maintain the HIPS Project/Vessel/Day/Line structure in the output, click the *Maintain PVD Hierarchy* check box.
4. Click **Next**.

HIPS To HOB

The Hydrographic Object Binary (HOB) file stores feature objects and associated attribute data. The HOB file can contain links to CARIS spatial objects.

HIPS to HOB (Step 3)



The *Export Soundings* option exports all soundings (subject to the sounding status and data binning settings).

1. Select the *Export Soundings* check box to export soundings to a CARIS map (see the next dialog box for the *Export Tracklines* and *Export Swaths* options).
2. To export soundings with a specific status flag, select any of the *Sounding Status* check boxes.

The *Data Binning* option divides the survey area into a grid for sounding selection. The *Bin Size* sets the size of the grid cells.

3. To apply data binning, select the *Data Binning* check box.
4. Type the size for the grid cell in the *Bin Size* field and select either *Metres* or *Feet*.
5. Select either *Shoal-* or *Deep-Biased* to export only the shoalest or deepest soundings, respectively, to the CARIS file.

Soundings can be selected for export from the total survey area or by processing each track line in sequence.

6. Select either the *By Area* or *By Line* options.

The *Export Track Lines* option connects a line along the centre of each swath, thus showing the track of the vessel along each survey line. It is not necessary to export soundings to use this option.

The *Export Swaths* command connects a set of lines (one per swath) along outermost port and starboard beams, thus showing the coverage of each ping. It is not necessary to export soundings to use this option.

7. Select the *Export Track Lines* and *Export Swaths* options.
8. Click **Next**.

HIPS to HOB (Step 4)

1. Click the **Browse** button to select a location for the HOB files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the HOB data and click **OK** to close the dialog box.

The file path is displayed in the *Output File* field.

3. Click **Next**.

HIPS To HTF

The HIPS and SIPS Export Wizard converts field sheet layer data and soundings to a Hydrographic Transfer Format (HTF) file.

Hydrographic Transfer Format

The HTF file is an ASCII file that can be opened in any text editor (such as Notepad). It consists of two sections: a header section and a sounding data record.

- The header contains general information about the survey.
- The sounding data record contains the depths and other relevant information.

The total number of fields generated in the sounding data record depends on the survey. At the start of the sounding data record is a Field Population Key that gives each field a 1 bit to show that a field is included, or a 0 bit to show if a field is not included.

For example: `[00001111100000000000]`

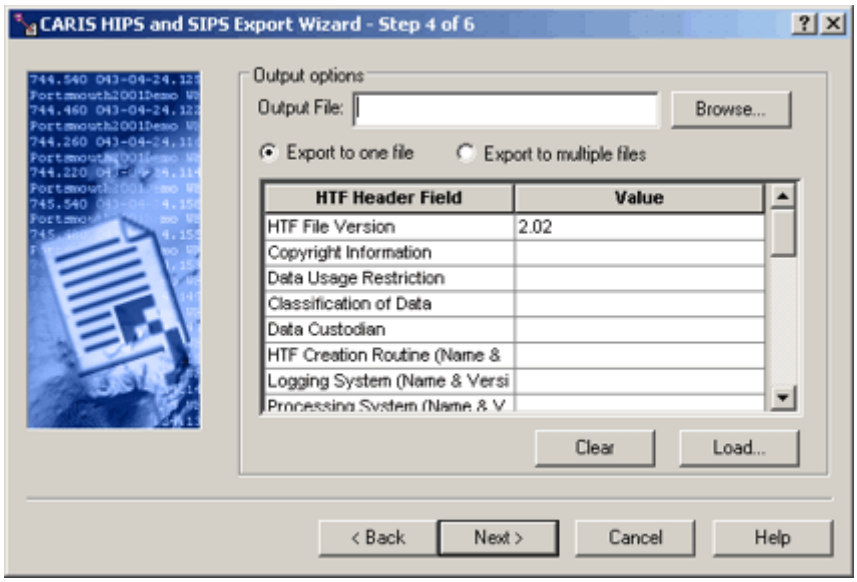
In the above Field Population Key, fields 5 to 9 contain data (hence the 1 bits), while fields 1 to 4 and 10 to 20 contain no data (hence the 0 bits). Fields without data are not written to the HTF file when it is generated. There are a total of 20 HTF fields in the sounding data record. Two additional fields are added by the Export Wizard for beam and profile number. Extra fields can be added, if needed. The HTF export automatically exports TPE depth and TPE position values if these are included in the track line.

The HTF file contains an ordered list of all possible fields and a description of each field.

Hydrographic Transfer Format Template

Sometimes the same header information can be used for multiple HTF files. The export wizard gives you the option of saving a header for use in another file. Headers are saved as HTFT files (Hydrographic Transfer Format Template). The file is in XML format that can be viewed in a text editor or an XML-compliant Internet browser.

HIPS to HTF

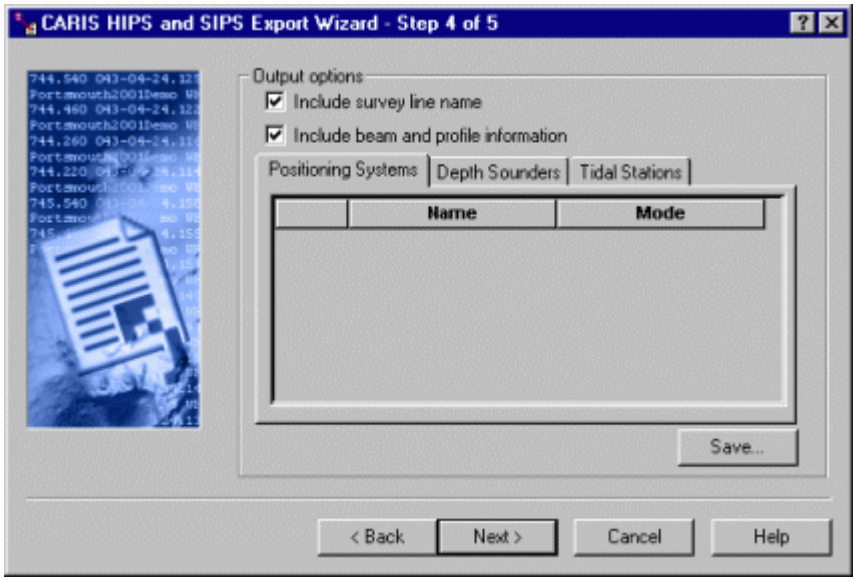


Enter the necessary header information. You can directly enter the data or load it from an HTFT (Hydrographic Transfer Format Template) file.

If you are exporting soundings, you can export all data to a single file or export data from each survey line to its own file. This option is not available for field sheet products.

1. Select an output location for the HTF file(s).
   - If you choose *Export to One File*, type a name and file path for the HTF file, or click **Browse** to select a location and name for the file.
   - If you choose *Export to Multiple Files*, click **Browse** to select a folder to save the HTF files. The files are automatically named according to Project_Vessel_Date_SurveyLine.
2. To enter new header information, click the *Value* field next to the *HTF Header Field* and type the data.
3. Click **Load** to open data from an existing HTFT file into the *Value* fields.
4. Click **Clear** to remove data from all fields, if needed.
5. Click **Next**.

HIPS to HTF



1. Select the *Include Survey Line Name* check box to include this information in the HTF file.
2. Select the *Include Beam and Profile Information* check box to include the beam and profile numbers in the HTF file.

If you entered values in the depth sounder, positioning system, and tidal station fields in the previous dialog box then the respective fields in these tabs are enabled. If you did not enter values, click **Back** and fill in the fields.

The Positioning Systems tab is for entering navigation systems information.

3. Type the name and model (and/or any additional information) of the system in the *Name* field.
4. Type the system used on the survey (GPS, DGPS, etc.).

The Depth Sounders tab is for entering sonar information.

5. Type the name and model of the sonar system used on the survey.
6. Type the mode of sonar (multibeam, single beam, multi transducer).

The Tidal Stations tab is for entering tide information.

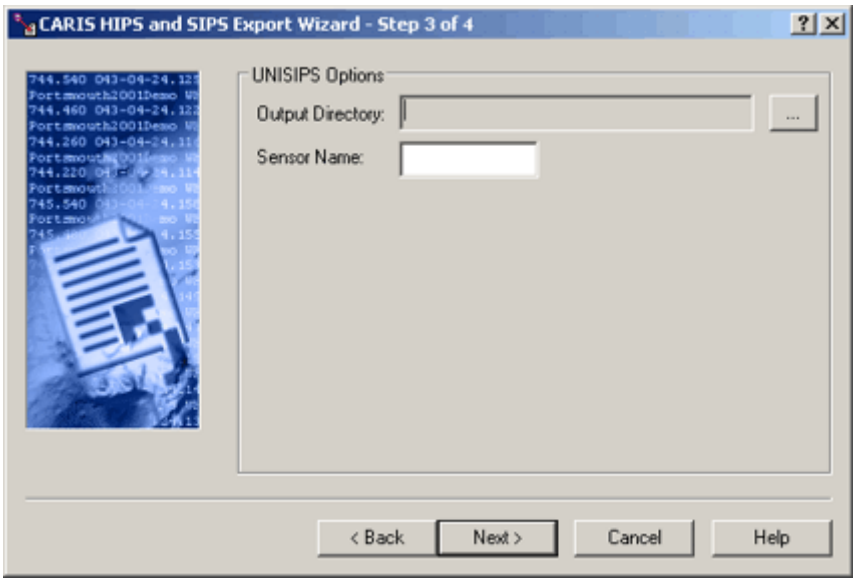
7. Type the name of a tidal station.
8. Type the location.
9. Type the method used to acquire data (predicted, observed, etc.)
10. Click **Save** to save information to a new or existing HTFT file.
11. Click **Next**.

SIPS To UNISIPS

Export SIPS data to a Unified Sonar Imaging Processing System (UNISIPS) format.

SIPS TO UNISIPS (Step 3)

Select a file to store the data.



1. Click **Browse** to select a location for the UNISIPS files.

The Browse for Folder dialog box is displayed.

2. Select a directory to store the UNISIPS data and click **OK** to close the dialog box. The UNISIPS file will have the same name as the track line with a .u file extension.
3. Type a *Sensor Name* in the field (up to 10 characters).

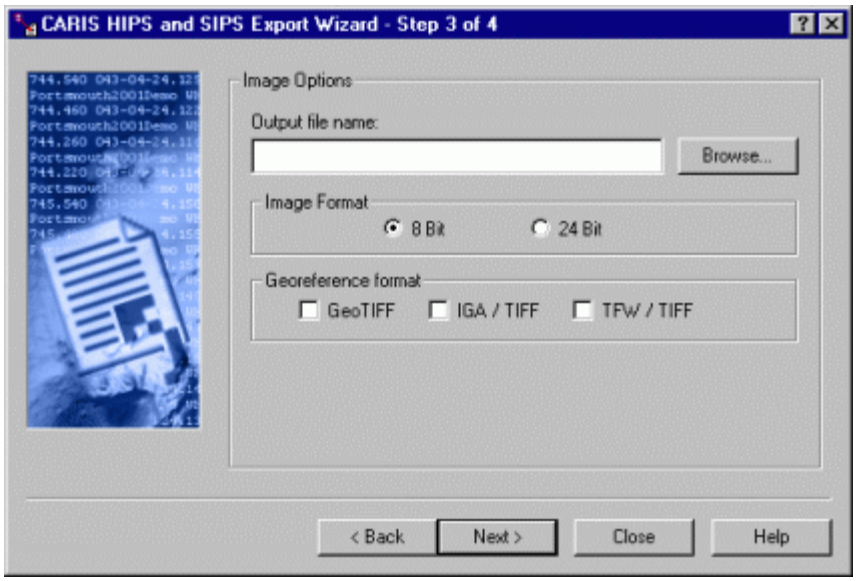
The file path is displayed in the *Output File* field.

4. Click **Next**.

Mosaic To Image

You can export a mosaic or group of mosaics to an image format for further processing in a CARIS product or other application.

Mosaic to Image (Step 3)



The field for the directory is labelled either as *Output File Name* or *Output Folder* if there is more than one mosaic for export.

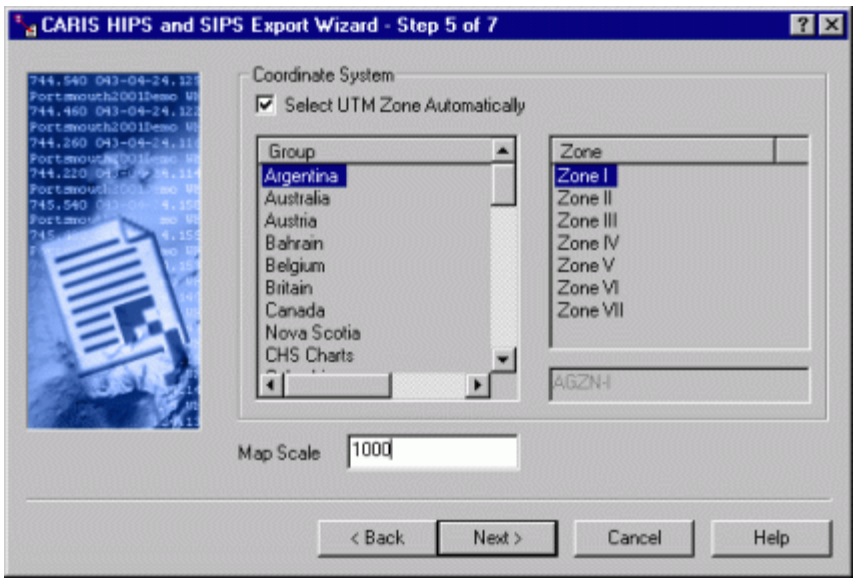
1. Click **Browse**, and select the file path and name of the image file. Or if you selected more than one mosaic, select the folder that contains the mosaics.
2. Select a colour format for the image.
   - *24 Bit (*16 million colours)
   - *8 Bit* (256 colours)

You can save the image as a TIFF file with either an IGA or a TFW georeferenced file, or both. You can save the image as a GeoTIFF image that records the georeference information directly in the .TIFF file.

3. Select a georeference format for the image:
   - *GeoTIFF*
   - *IGA/TIFF*
   - *TFW/TIFF*
4. Click **Next**.

Projection

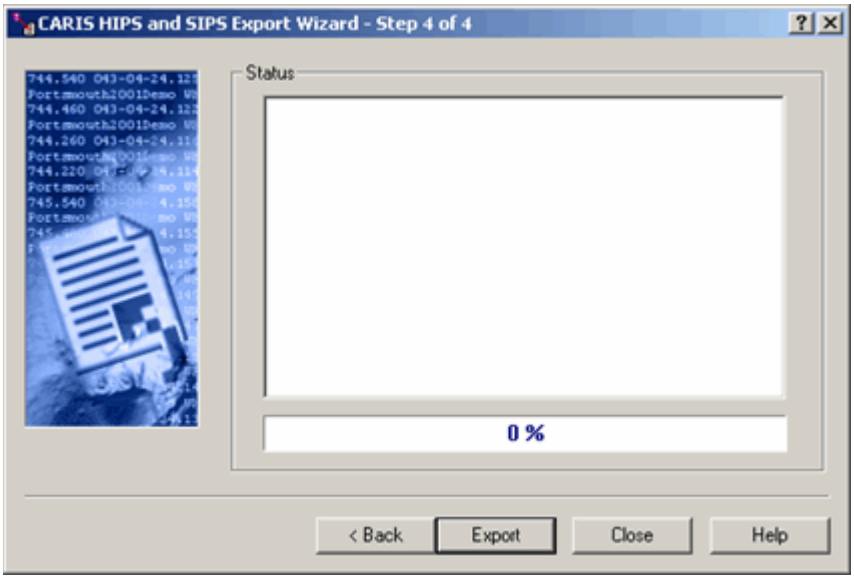
All sounding data is stored in HIPS and SIPS as unprojected longitude and latitude. This dialog selects the appropriate coordinate system to reference the soundings. All soundings will be transformed to this coordinate system as they are exported.



1. Choose a coordinate system by selecting a Projection Group from the list.
2. Select a Zone associated with the Projection Group (the Key field is filled when the Zone is selected).
3. Click Next.

Export

This dialog box shows the percentage of data exported and the status of the export process.



1. Click **Export**.

A progress indicator shows the percentage of files converted to the selected format.

A

HIPS Status Flags

Status Flags and Values

Within HIPS and SIPS data structure each sensor data value has a series of status flags. Each profile in the depth and side scan formats also has status flags, and each survey line has status flags. The status flag settings are determined during data cleaning and processing. They are used to control the generation of new data layers such as BASE surfaces, tiles, and selected soundings.

Status Flag	Value
Accepted	The default status of sounding.
Rejected	True if the sounding is rejected for any reason.
Examined	Indicates that the sounding was verified.
Designated	Indicates that the sounding is the shoalest in a cluster of soundings.
Outstanding	Indicates that the sounding needs further examination.
Rejected by Swath Editor or Single Beam Editor	True if sounding was rejected in the Swath Editor or Single Beam Editor, or by the swath or single beam filters.
Rejected by Hydrographer	True for soundings rejected manually by the hydrographer within the Subset Editor.
Rejected by Surface Cleaning	True if sounding was rejected during surface cleaning
Rejected by Depth Gate	True for soundings rejected by the depth gate during conversion.
Rejected by Disabled Beam	True for soundings rejected at conversion time because a disabled beam flag in the VCF or due to a pre-flagged bad sounding in the original raw data format.
Rejected by TPE	True for soundings rejected during Total Propagated Error filtering.
Rejected by surface filter.	True for soundings rejected during surface filtering.
Rejected by Auto Classification	True if the sounding fails the auto classification tests filter contained in the HIPS v4.3.3 subset mode. This flag is not used in v5.x software.
Rejected by Bad Navigation	True if the profile was rejected for bad navigation.
Rejected by Bad Gyro	True if the profile was rejected for bad gyro.
Rejected by Bad Heave	True if the profile was rejected for bad heave.
Rejected by Bad Pitch	True if the profile was rejected for bad pitch.
Rejected by Bad Roll	True if the profile was rejected for bad roll.

Status Flag	Value
Rejected by Bad Tide	True if the profile was rejected for bad tide.

B

Generic Data Parser

Introduction

HIPS can convert almost any single beam ASCII file through the Generic Data Parser. The parser can also be used to add or replace any sensor data in an existing project, except swath and sweep bathymetry and side scan imagery.

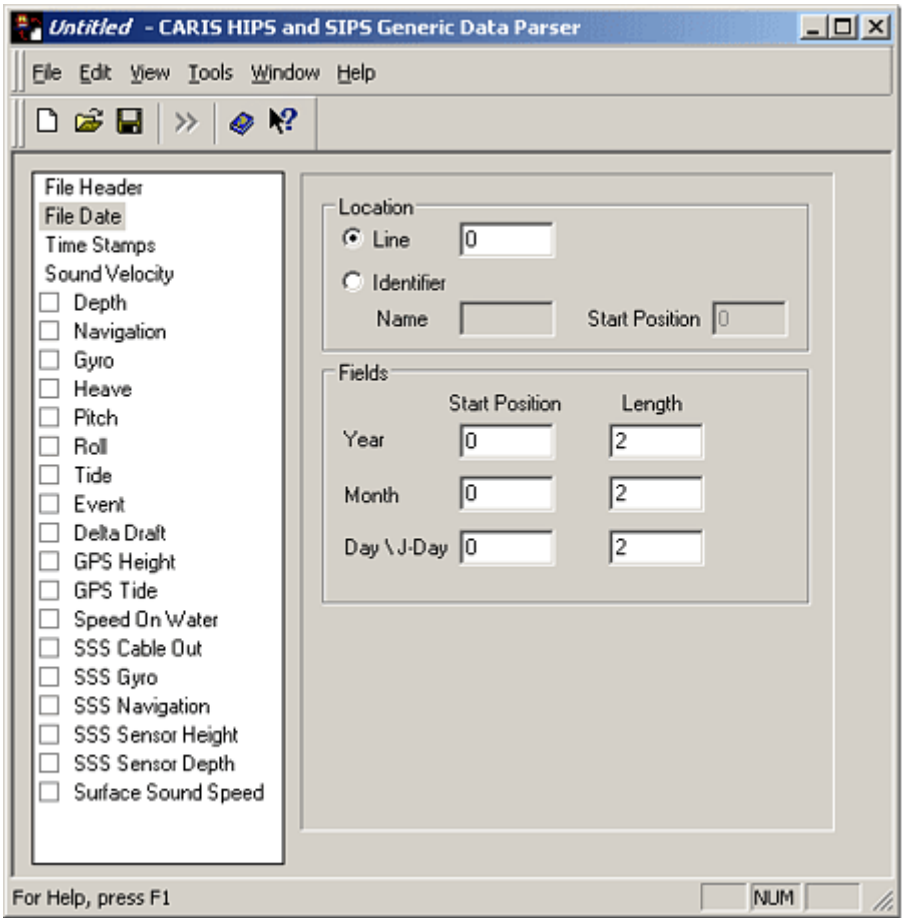
For example, if high-precision positions were obtained from a source other than the original raw data files converted into HIPS, this data could be loaded into an existing project to replace the original navigation data. The new position data must have a time stamp and be in ASCII format.

The Generic Data Parser is a separate program launched from the main HIPS and SIPS interface.

Menu	Import > Generic Data Parser
Tool	

1. Select the Generic Data Parser command.

The Generic Data Parser interface is displayed, as shown in the example below.



To open raw data to be converted with the parser,

1. Select Open Raw Data from the File menu.
2. Browse to select the *.raw or *.txt data file you want to convert.

The Raw Data window will open to display the data. This window can be sized and repositioned to the top, bottom, right or left sides of the interface.

Data Parsing Components

The parser must be told where each piece of information is located in the ASCII text file. This set of parameters is stored in a configuration file that can be saved, retrieved or modified at any time. These configuration files are stored in the …\Hips\Session folder and have a .par extension.

The location of each type of data in the text file is determined using either the free form or fixed column method. In the raw data file the data fields are separated by delimiters, and lines of data are differentiated by identifiers.

Free Form

The free form method is used when data items (separated by delimiters) have fields of varying lengths. In this case, you specify a field number to show where a data item is located in the text file. The field numbering starts at one (1) and is counted from the first character following one or more delimiters.

For example, in the input string 301.12, 45, 5.66 — field number three (3) indicates the start of string 5.66.

Quotation marks can be used to offset a text string, for example, “312, 45, 5.66” is treated a single string field (regardless of any characters or spaces inside the string).

The following characters are used as delimiters by the parser:

- tab
- space
- single quote (‘)
- vertical bar (|)
- comma (,)
- semicolon (;)

These delimiters are replaced with spaces when the parser reads the file.

Fixed Column

This method specifies the absolute location of a data item in the text file. The location starts at one (1). This method is used when the location of data items does not change and where there is no

obvious separation between fields. You must supply the length of the field when using the fixed form method.

Identifier

When data items are stored in separate text lines, the program must differentiate between types of input. This is done with an identifier, which is a short text string whose location is specified as an absolute value. If an identifier is specified for a data item, the input line is tested for the item's presence to make sure that it can be extracted.

For example, some depth data lines in a Hypack data file start with the string EC2. In this case, you specify the identifier as EC2 at the location one (1). You can include spaces and other symbols in the identifier, as long as the symbols are not classified as belonging to any of the delimiters.

The use of identifiers is optional for any data item. In this case, the program extracts the data item on all input text lines.

Multiplier/Offset

Once a value has been extracted from the input text line, additional operations may be needed to transform it to its final value.

This may be necessary because:

- The value is stored with an implied decimal point (for example $1/100^{th}$ of a second).
- The value is stored in a different unit (for example, decimal radians instead of degrees).
- The sign of the value is opposite of what HIPS uses.
- The value is stored with an implied offset.

You may specify additional mathematical operations in terms of a Multiplier and an Offset. The actual formula used is:

```
final value = (extracted value x Multiplier)
+ Offset
```

Pseudo Time

The pseudo time stamp is a user-determined time sequence that overrides the Data Parser's time stamp requirement for record conversion. By default, the time is set to midnight at the current date, but this can be changed. An increment value attaches a new time to each record. Make sure that the pseudo time stamps are comparable with the types of data you are importing.

If you are using this feature, the Time Stamps panel information must still be completed, but the pseudo time stamp overrides this information.

Configuration File

The configuration file is a text that contains the settings for converting data through the Generic Data Parser. All configuration files have a PAR extension.

"Open a Configuration File" on page 631

"New Configuration Files" on page 632

"Save a Configuration File" on page 632

Open a Configuration File

You can open an existing configuration file for editing and then convert text files with the Generic Data Parser wizard.

Menu	File > Open
Tool	
Key	<Ctrl+O>

1. Select the Open File command.

The Open dialog box is displayed.

2. Select the configuration file you want to open.
3. Click **Open**.

The configuration file is now ready for editing or for proceeding directly to converting raw data files.

The data types included in the current configuration are shown by a check mark.

4. Optional: Select a data type and modify any parameters, as needed (see "Data Types" on page 635).
5. Optional: Select or de-select a data type by clicking the check box to include or exclude data types during conversion.

New Configuration Files

You can select data types and enter values for a new configuration file.

1. Select the New File command.
2. Select a data type and enter values for the parameters.

> Remember: You must complete each section before selecting the next data type.

Menu	File > New
Tool	
Key	<Ctrl+N>

3. To reset any changes to the data, select the Reset Selected Sensor command.
4. Select the check box to include the associated data during conversion.
5. Repeat Step 2 and Step 3 until all necessary data types have been added to the new configuration file.
6. Select a Save/Save As command.

Menu	Edit > Reset Selected Sensor
Pop-up	Reset

The Save As dialog box is displayed.

7. Type a name for the configuration file and select a directory for the file.
8. Click **Save**.

Save a Configuration File

9. When finished, select the Save or Save As commands; or start the conversion wizard (see “GENERIC DATA PARSER WIZARD” ON PAGE 641).

If you selected the Save or Save As commands, then Save As dialog box is displayed.

10. Enter a name for the file and select a location to save it.
11. Click a Save command.

Menu	File > Save/Save As
Tool	
Key	<Ctrl+S>

Raw Data Files

Menu	File > Open Raw Data

You can open a text file and find the exact column positions of the data you want parsed.

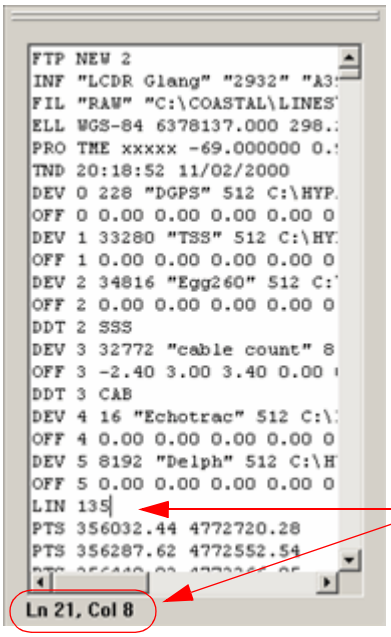
1. Select the Open Raw Data command.

The Open dialog box is displayed.

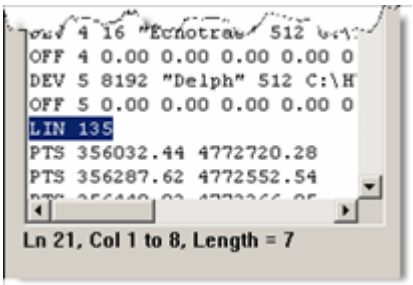
2. Select a text file and click **Open**.

The file is opened in the Editor.

When you position the cursor in the raw data file, the cursor location is displayed below the window.



When a section of the file is highlighted, the length (including characters and spaces) of the highlighted area is shown.



3. Select a data type from the list so the associated parameter fields are displayed in the parser.
4. Select the appropriate area in the text file by either placing the cursor in the file or highlighting a section of it.

5. Enter the raw data file information in the parameters area (see “DATA TYPES” ON PAGE 635).
6. Continue with Steps 3 to 5 until all information has been collected for the configuration file.

Close files

Menu	File > Close Raw Data

7. To close files without saving information, select the Close Raw Data command.

Data Types

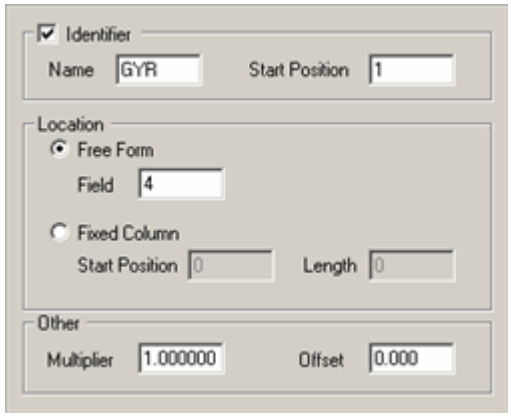
The data types are listed on the left side of the interface.

If the check box beside the data type is selected then that data is available for conversion. Each data type has a set of parameters that are displayed when it is selected. These parameters determine what parts of the text file are parsed for the configuration file.

To view the parameters:

1. Select a data item so it is highlighted.

The parameters for the data type are displayed in the Parser.



2. Enter data as needed.

When you select another data type, the previous type is now checked (if it has a check box). This means that when the text file is parsed, that data is included in the conversion. If you do not want to include the data in the conversion, make sure the check box is cleared.

Reset selection

Menu	Edit > Reset Selected Sensor

3. To remove changed parameters from a data item, select the Reset Selected Sensor command.

The parameters are reset to the default positions.

You must complete all the fields in each section before selecting another data type.

File Header

This section indicates the number of lines in the header. These lines are skipped when the text file is parsed. The default value is zero.

1. Type the number of lines used for the file header.

File Date

1. Select either a line number or Identifier so the file data can located.
   - If you selected line number, type the number in the text box provided.
   - If you selected identifier, type a text string for the identifier in the *Name* field and the location on the line where the date begins.
2. Define the location of the date values within the field.
   - If the length of the year field is two (2), then the converter automatically assigns the year a value between 1951 and 2050.
   - If the length of the day field is three (3), the converter assumes the extracted value represents a Julian date.

Time Stamp

Each line must contain a time stamp. The various components in the time stamp are specified in terms of absolute positions in relation to the start of the time stamp string.

1. Select the location of the time stamp as a Fixed Column or Free Form position.
2. Include year/month/day information:
   - Select the *Yes* check box so the year/month/day fields are enabled.
   - Type a position for the time stamp and the length of the year parameter.
3. To add hour and minute, select the appropriate check boxes and type the position and length information.
4. Enter the start and length values for the seconds parameters.
5. Enter any Multiplier/Offset values.

Sound Velocity

You can enter a sound velocity value directly or parse it from the input file. When converting single beam depth data, the sound velocity value can be used to recover the raw travel time from the recorded depths. This raw travel time can then be used to perform full sound velocity corrections on the data during post processing.

1. Select the position for the parser to start reading data by selecting either the *Line Number* or Identifier option.
2. Enter a line number to start reading the sound velocity data, or enter an identifier and start position for the data.
3. Select the location of the SVP data as a Fixed Column or Free Form position.
4. Enter the relevant free form or column position.
5. Enter any Multiplier/Offset values.
6. Type a default sound velocity value if you want to enter the sound velocity value directly.

Depth

For each depth, indicate the location of the sounding data and whether any additional operations are needed.

1. Select the Identifier check box if the information is parsed from a specific line in the text file.
2. Type the name of the identifier and the start position for reading the depth data.
3. Select the dual frequency check box if the text file contains secondary single beam data that must also be converted.

The primary and secondary tabs contain identical fields. Enter data in one or both sections, as needed.

4. Select either Free Form or Fixed Column.
   - If *Free Form*, enter the field number where the data is located.
   - If *Fixed Column*, enter the start and length of data in the text file.
5. Enter any Multiplier/Offset values, as needed.
6. Select the *Compute Raw Travel Time* check box, if you want to calculate the raw travel time from the sonar to the seabed.
7. Select the Pseudo Time check box.
8. Type a *Start* date and type an *Increment* value (in seconds).

Navigation

The converter supports reading both ground and geographic coordinates. For geographic Latitude and Longitude values, negative values are supported as well as E/W and N/S tags.

1. Select the Identifier check box if a text string is used to identify navigation data in the text file.
2. Enter the identifier and start position for the identifier.
3. Select the type of coordinates used: geographic or ground.

The content of tabs below this section is determined by type of coordinates selected.

- If *Geographic* is selected, the Latitude and Longitude tabs are displayed.
- If *Ground* is selected, the North and East tabs are displayed.

4. Complete the following fields for the Latitude/Longitude or the North/East tabs:
   - *Free Form*: The free form method used to locate navigation data.

- *Field*: The field where the data is located, if you selected the Free Form option.
- *Fixed Column*: The fixed position method used to locate data.
- *Position*: The absolute location of the latitude or longitude data if you selected the Fixed Column option.
- *Degrees*: These fields contain the start of the coordinate field, the length of the field and any multiplier and offset values.
- *Min*: This check box enables the Start, Length, Multiplier/Offset fields when selected (geographic coordinates only).
- *Sec*: This check box enables the Start, Length, Multiplier/ Offset fields when selected (geographic coordinates only).
- *Tags*: A hemisphere tag (N/S/E/W) is parsed with the data when this option is checked.
- *Position*: Position of the tag in the text file.

5. [Optional] Select the Pseudo Time check box.
6. Type a *Start* date and type an *Increment* value (in seconds).

Attitude/Navigation/Tide

This section includes the following data types:

- Gyro
- Heave
- Pitch
- Roll
- Tide
- GPS Height
- GPS Tide
- Speed on Water
- SSS Cable Out
- SSS Gyro
- SSS Navigation

All the fields in these sections are identical.

1. Select the *Identifier* check box, if the data is going to be associated with an Identifier.
2. Enter an identifier and start position for the data.
3. Select the location of the data as a Fixed Column or Free Form position.
4. Enter the relevant free form or column position.

5. Enter any Multiplier/Offset values.

Event

Event marks are sometimes generated automatically by the logging system to indicate that a certain amount of time has elapsed, or a certain distance has been travelled. The event marks can be displayed in the Single Beam Editor. In HIPS, event mark records consist of one or more unique event numbers and textual comments. In this dialog box, you must specify where to look for the event mark number within the data record.

1. Select the *Identifier* check box, if the data is going to be associated with an Identifier.
2. Enter an identifier and start position for the data.
3. Select the location of the data as a Fixed Column or Free Form position.
4. Enter the relevant free form or column position.
5. If the Events marks contain a textual description, check the *Comments* check box.
6. Type the *Start Position* of the comments in the text file.
7. Type the *Length* of the comment.
8. Select the *Time Stamp* check box to include the time stamp that is included with the event. If this option is not checked, the time stamp parsed from the file is used.

Generic Data Parser Wizard

Single beam data or sensor data from a text file is converted to HIPS/SIPS format with a six-step wizard opened from the Generic Data Parser.

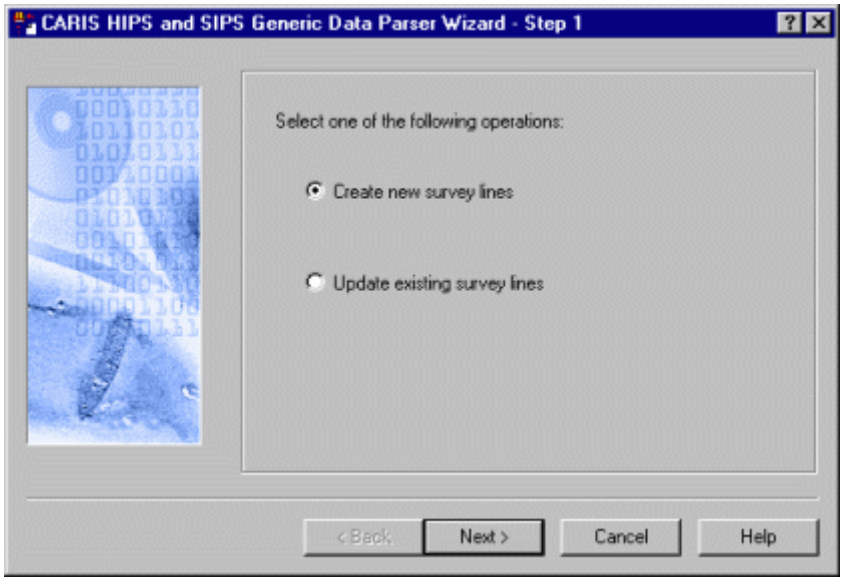
> The appropriate configuration file must be open in the Parser before data can be converted.

Open the wizard

Menu	Tools > Run
Tool	

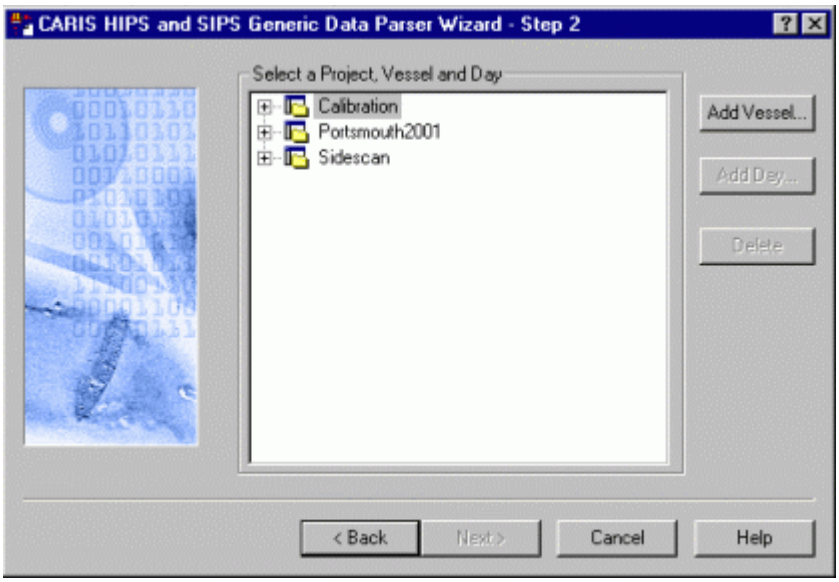
1. Make sure that a configuration file is open in the Parser.
2. Select the Run command.

The GDP Wizard - Step 1 dialog box is displayed.


CARIS HIPS and SIPS Generic Data Parser Wizard - Step 1
Select one of the following operations:
Create new survey lines
Update existing survey lines
< Back
Next >
Cancel
Help

3. Select one of the following operations:
   - Convert data into new CARIS HIPS/SIPS survey lines.
   - Convert data into existing CARIS HIPS/SIPS survey lines.
4. Click **Next**.

The GDP Wizard - Step 2 dialog box is displayed.



You must select a Project-Vessel-Day directory to save the converted data if creating new survey lines, or select an existing survey line if you are updating existing data.

5. Click a Project file tree so it is expanded and the Day folder or survey line is visible.

If you want to add Vessel or Day folders or delete a folder, follow Steps 5 to 9, otherwise go to Step 10.

6. To add a vessel folder, make sure the project directory is highlighted and click **Add Vessel**.

The Available Vessels dialog box is displayed.

7. Click a vessel folder and click **OK**.

The new Vessel folder is displayed in the selected project directory.

8. To add a day folder, click the Vessel folder and click **Add Day**.

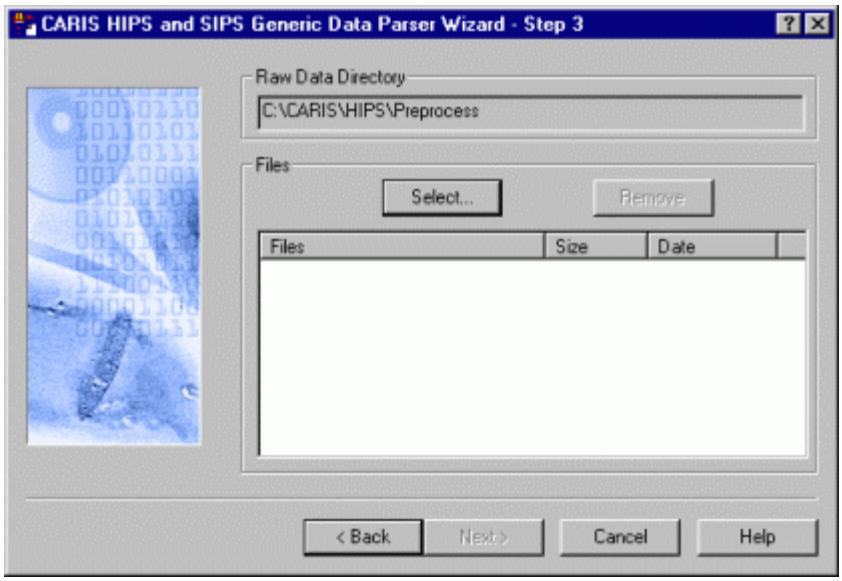
The Calendar dialog box is displayed.

9. Select a year/month/day from the calendar, and click **OK**.

A Day folder with the selected date is stored in the Vessel folder.

10. To remove an empty Day or Vessel folder from the wizard, highlight the folder and click **Delete**. A Vessel or Day folder cannot be deleted using the **Delete** button, if its branch of the Project already has data converted into it.
11. Click the Day folder or survey line so it is highlighted.
12. Click **Next**.

The GDP Wizard - Step 3 dialog box is displayed.



In this dialog box you select the text files you want to convert.

13. If the file path in the *Raw Data Directory* box is correct, click **Select**.

The Select Files dialog box is displayed.

14. Double-click a folder to access the text files.

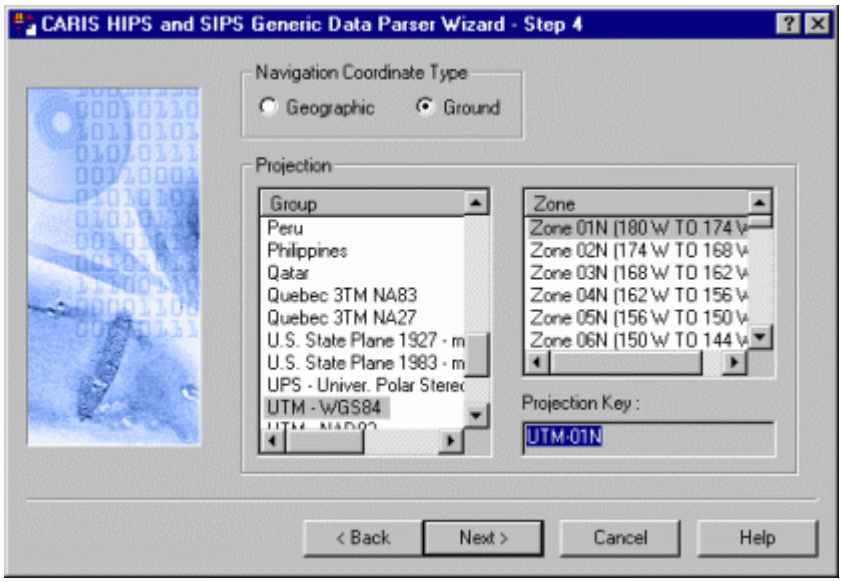
The files are displayed.

15. Select the files so they are highlighted.
16. Click **Open**.

The selected files are displayed in the wizard dialog box.

17. To remove files, select the files and click **Remove**.
18. Click **Next**.

The GDP Wizard - Step 4 dialog box is displayed.



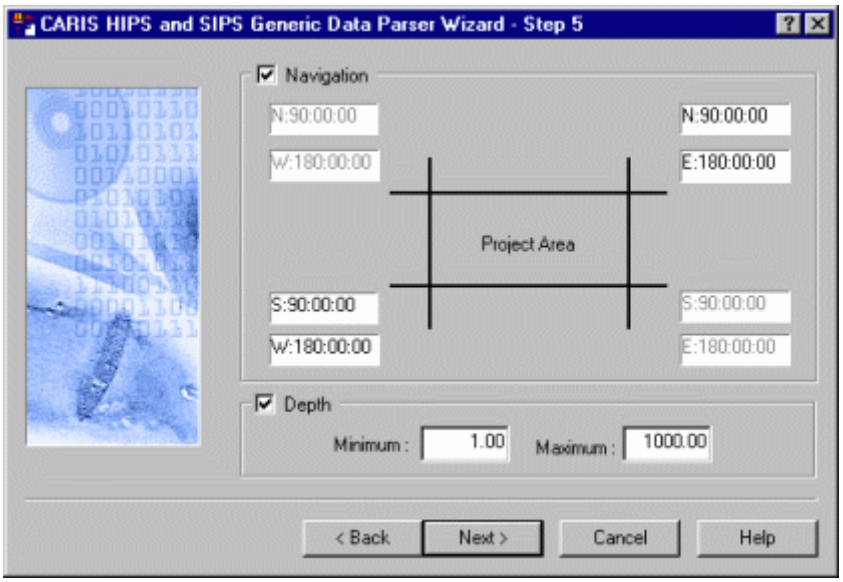
In this dialog box you must indicate the type of coordinate system that was used for the Navigation data during data logging.

19. Select one of the following options:
    - *Geographic*: navigation data is recorded as latitude and longitude coordinates.
    - *Ground*: navigation data is recorded as eastings and northings.

If you select *Ground*, complete the following steps:

20. Select a projection group from the *Group* list.
21. Select a zone.
22. Click **Next**.

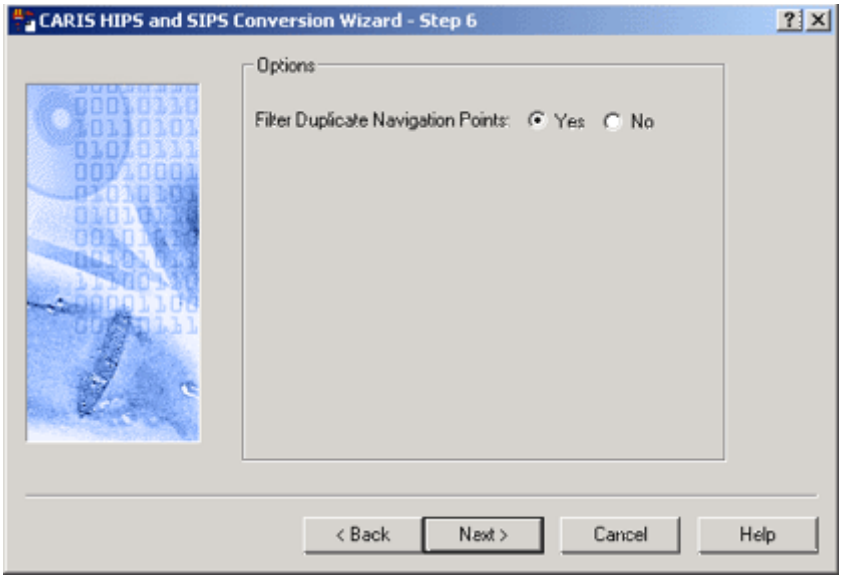
The GDP Wizard- Step 5 dialog box is displayed.



In this dialog box you can reject extremely large errors in the recorded navigation and depth data. Navigation and depth filters can be applied during conversion with the following steps:

23. Click the *Navigation* check box to enable the navigation filter.
24. Enter coordinates in the boxes to define the area:
25. Click the *Depth* check box to enable the depth filter.
26. Enter the minimum acceptable depth.
27. Enter a maximum acceptable depth.
28. Click **Next**.

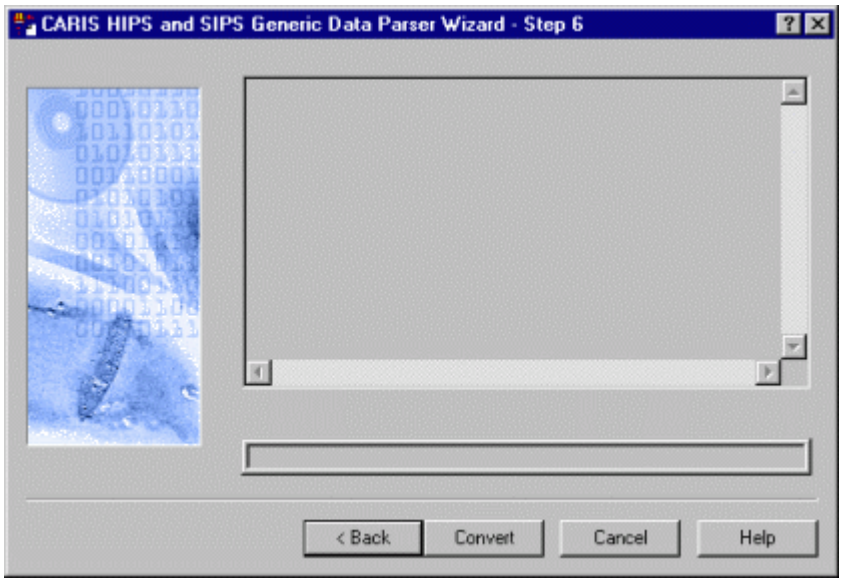
The GDP Wizard - Step 6 dialog box is displayed.



Duplicate navigation points are recorded when the vessel is stationary. The *Filter Duplicate Navigation Points* option gives you to option to remove the duplicate points during import or to include them in the HIPS file (therefore representing the vessel as stationary).

29. Select *Yes* or *No* to for the *Filter Duplicate Navigation Points* option.
30. Click **Next**.

The GDP Wizard - Step 7 dialog box is displayed.



In this dialog box you convert the text data to HIPS/SIPS format.

31. Click the **Convert** button.
32. A progress indicator bar along the bottom of the box is activated as the files are converted.
33. A text box displays the results of the conversion process.

The HIPS files are saved in the directory you selected in the second dialog box.

34. Click **Finish** to close the wizard, or click **Restart** to return to the first dialog box of the wizard.

C

Tide File Formats

Overview

HIPS supports a standard Canadian Hydrographic Service tide format called COWLIS as well as two NOS/NOAA formats. HIPS also supports a basic format consisting of just the required date, time, and tide values.

HIPS does not use predicted tide tables.

The time zone of the tidal observation data must match the time zone of the bathymetry to which it will be applied.

Tide files (*.tid)

TID files are ASCII files consisting of date, time, and tide values collected from a single tide station. These files can be created or edited in a text editor (such as Notepad), or in the Tide Editor.

The following tide data file formats are supported by HIPS:

- CHS Cowlis
- CHS TMS 50
- CHS TMS 1000
- NOS Preliminary Tide Data
- NOS Final Tide Data

Basic Tide Format

The basic tide format used by HIPS begins with a single line containing at least 8 dash characters, followed by tidal observation records consisting of date, time, and tide. Here is a simple example:

```
--------
2000/11/01 00:00   0.522
2000/11/01 01:00   0.317
2000/11/01 02:00   0.395
2000/11/01 03:00   0.751
2000/11/01 04:00   1.296
2000/11/01 05:00   1.848
2000/11/01 06:00   2.339
2000/11/01 07:00   2.589
```

For information on using the Tide Editor, see "EDITING TIDE DATA" ON PAGE 340.

COWLIS

```
---------------------- LIBEX file dump ----------------------
------------------------ STN?????.OBS ------------------------
------------------ Time Zone:  GMT ( 0.0) ------------------

--------------------- Invariant Fields ---------------------
Name              Type Size    Units Value
------------------------------------------------------------
station_id        CHAR    5
station_name      CHAR   16          Rimouski, Quebec
data_product      CHAR    3          TMS1000
start_time        INTG    4  seconds 93/06/29 18:45:00
end_time          INTG    4  seconds 91/11/16 20:45:00
file_date         INTG    4  seconds 95/12/20 15:24:24
max_water_level   REAL    4   metres    1.640
min_water_level   REAL    4   metres    0.050
------------------------------------------------------------

---------------------- Variant Fields ----------------------
Name              Type Size    Units
------------------------------------------------------------
time              INTG    4  seconds
water_level       REAL    4   metres
std_dev           REAL    4   metres
------------------------------------------------------------
1993/06/29 18:45:00   1.95  0.030
1993/06/29 19:00:00   1.84  0.030
1993/06/29 19:15:00   1.74  0.030
1993/06/29 19:30:00   1.63  0.030
1993/06/29 19:45:00   1.55  0.030
1993/06/29 20:00:00   1.48  0.030
1993/06/29 20:15:00   1.41  0.030
1993/06/29 20:30:00   1.37  0.030
1993/06/29 20:45:00   1.34  0.030
-------------------- End Of LIBEX File --------------------
```

NOS Preliminary Tide Data

```
************************ PRELIMINARY DATA ***************************

     THIS RAW DATA HAS NOT BEEN SUBJECT TO THE NATIONAL OCEAN SERVICE'S
     QUALITY CONTROL OR QUALITY ASSURANCE PROCEDURES AND DOES NOT MEET
     THE CRITERIA AND STANDARDS OF OFFICIAL NATIONAL OCEAN SERVICE
     DATA.  IT IS RELEASED FOR LIMITED PUBLIC USE AS PRELIMINARY DATA TO
     BE USED ONLY WITH APPROPRIATE CAUTION.

************************ PRELIMINARY DATA ***************************

      Water Level -  Acoustic (A1)
      Station    --  Unique seven character identifier for the station
      DCP        --  A one character identifier for the data collection
                     platform at a station
      SE         --  A two character identifier for the data sensor
      Date Time  --  Date and time the data were collected by the DCP
      WL_Value   --  Water level height
      Sigma      --  Standard deviation of 1 second samples used to
                     compute the water level height
      Out        --  Count of number of samples that fall outside a
                     3-sigma band about the mean
      Flat       --  A flag that when set to 1 indicates that the flat
                     tolerance limit was exceeded
      ROFC       --  A flag that when set to 1 indicates that the rate
                     of change tolerance limit was exceeded
      Temp       --  A flag that when set to 1 indicates that the
                     temperature difference tolerance limit was exceeded
      Limit      --  A flag that when set to 1 indicates that either the
                     maximum or minimum expected water level height limit
                     was exceeded

      Data are in meters above MLLW
      Times are on UTC (GMT)

9453220 1 Yakutat, AK  from  19990614 to 19990618
  ------------------------------------------------------------------------
Station  DCP  SE  Date       Time   WL_Value   Sigma  Out Flat RofC Temp
Limit

9453220   1   A1  1999/06/14 00:00     1.881   0.014   0    0    0    0    0
9453220   1   A1  1999/06/14 00:06     1.814   0.025   0    0    0    0    0
9453220   1   A1  1999/06/14 00:12     1.746   0.018   0    0    0    0    0
9453220   1   A1  1999/06/14 00:18     1.692   0.014   0    0    0    0    0
9453220   1   A1  1999/06/14 00:24     1.632   0.013   0    0    0    0    0
9453220   1   A1  1999/06/14 00:30     1.560   0.015   0    0    0    0    0
  ------------------------------------------------------------------------
```

NOS Final Tide

```
 NOS SIX MINUTE WATER LEVEL HEIGHTS

      Data are verified

      Station   --   Unique seven character identifier for the station
      Date Time --   Date and time the data were collected by the DCP
      WL_Value  --   Water level height
      Sigma     --   Standard deviation of 1 second samples used to
                     compute the water level height
      Infer     --   A flag that indicates that the water level value
                     has been inferred.
      Flat      --   A flag that when set to 1 indicates that the flat
                     tolerance limit was exceeded
      RofC      --   A flag that when set to 1 indicates that the rate
                     of change tolerance limit was exceeded
      T_Flag    --   A flag that when set to 1 indicates that the
                     temperature difference tolerance limit was exceeded

      Data are in meters above MLLW
      Times are on UTC (GMT)

9452210 Juneau, Gastineau Channel, Stephens Pass, AK, USA  from  19980525 to
19980530
  ------------------------------------------------------------------------
Station Date       Time    WL_Value   Sigma  Infer Flat RofC T_Flag

9452210 1998/05/25 00:00     2.473     0.019     0    0    0     0
9452210 1998/05/25 00:06     2.330     0.022     0    0    0     0
9452210 1998/05/25 00:12     2.200     0.019     0    0    0     0
9452210 1998/05/25 00:18     2.064     0.020     0    0    0     0
9452210 1998/05/25 00:24     1.938     0.019     0    0    0     0
  ------------------------------------------------------------------------
```

Tide Zone Definition Files

The following is an example of a Zone Definition File (.zdf).

```
[ZONE_DEF_VERSION_3]

[ZONE]
PIS1,10
43.069551, -70.687251
43.061704, -70.717396
43.067896, -70.717241
43.071497, -70.712564
43.072329, -70.710498
43.075112, -70.710772
43.078966, -70.71016
43.081576, -70.708724
43.082372, -70.690817
43.069551, -70.687251

[ZONE]
PIS2,11
43.08703, -70.720529
43.07693, -70.724628
43.072765, -70.724696
43.071497, -70.712564
43.072329, -70.710498
43.075112, -70.710772
43.078966, -70.71016
43.081576, -70.708724
43.083356, -70.715428
43.086439, -70.715696
43.08703, -70.720529

[TIDE_ZONE]
PIS1,419870,PRIM,-12,1.06,0.0,0.2
PIS2,419871,PRIM,-12,1.05,0.0,0.2

[TIDE_STATION]
419870,43.067896,-70.717241,2.0,0.1,D:\HIPS\Tide\419870.tid
419871,43.072329,-70.710498,1.0,0.15,D:\HIPS\Tide\419870.tid

[TIDE_AVERAGE]
PIS1,419870,419871
PIS2,419870,419871

[OPTIONS]
Outage, 600
Interval, 10
```

Tide Zone Data

The Zone Definition File has the following components:

- The mandatory header section must consist of this text string: [ZONE_DEF_VERSION_3]. This specifies the version of the Zone Definition File.
- The section labelled [ZONE] contains the name of the zone(s) and the coordinates that define the boundary of the zone(s). All tide zone files must contain this section.

  There are two fields in the first line:

  - <Zone Label> contains name of the zone (the name must not be longer than 256 characters).

- number of points that define the boundary of the zone. (These are listed starting in the next line of the section.)

The remaining lines in this section list the geographic coordinates of these points, (latitude, longitude) in decimal degrees.

The last line in the list must contain the same coordinates as the first line to close the polygon.

If there is more than one zone being defined, this section is repeated with the definition information for each zone.

- The section labelled [TIDE_STATION] contains the definition of the tide stations collecting the tide observations. There are six fields in each record of this section:
  - *station label* (the name of the tidal station) - The name must not be longer than 256 characters.
  - *longitude* in decimal degrees - Valid ranges are from -180.0 to +180.0. The negative range represents the Western Hemisphere.
  - *latitude* in decimal degrees - Valid ranges are from -90.0 to +90.0. The negative range represents the Southern Hemisphere.
  - *max amplitude* (in metres) expected from the tidal station - This value will be used to compute a scaling factor during the Tidal TPE computation.
  - *uncertainty* error value associated with the data from the station (in metres)
  - [optional] full file path and name of the tide file associated with the station ID - If this is omitted, HIPS will look for an observation file in the same directory as the *.zdf, with the file name <Station_Label>.tid.
- The section labelled [TIDE_ZONE] contains specific attributes that define a zone's tidal parameters. There are seven fields in this section:
  - tide *zone label* - this must match the name of the zone(s) defined in the [ZONE] section.
  - tide *station label* - name of the tidal station associated with this entry.
  - *priority* designation of the tidal station entry: either the primary (PRIM), secondary (SEC), tertiary (TER) or preliminary (PRELIM) station for this zone.
  - *time correction* - the tide offset for this zone (in minutes)
  - *range correction* - a multiplier (for example, 1.06) used to scale the tidal value read from the observation file

- *tidal shift* - a number in metres added to the values after range correction has been applied.
- *uncertainty* value for the tidal zone (in metres) used to compute the Tidal Zone Error.

- The section labelled [TIDE_AVERAGE] assigns tide stations to zones for tide-weighted averaging. Each record has at least two station fields because each zone can be assigned one or more tide stations for the weighted average calculation.

  The section has the following fields:

  - tide *zone label* - the zone (as defined in the [ZONE] section) to which the averaging should be applied
  - tide *station label*, (as defined in the [TIDE_STATION] section). Enter as many as needed for averaging.

- The last section labelled [OPTIONS] contains options used during tide zoning or tide averaging.
  - The limit value field contains the outage time in seconds before switching from the current station to the next priority station
  - final interval in seconds of tide data loaded into the track line.

For information on applying the data in a ZDF to survey lines see "LOAD TIDE" ON PAGE 215.

D

Contact File Formats

Introduction

Contacts in SIPS are stored in ASCII files in each line directory. SIPS gathers all of these files together and produces a single set of output files that represent all contacts from all of the survey lines. The SIPS Contact export function from SIPS produces four files:

- **filename_ContactLine.txt** contains information identifying the survey line and indexes to the other files.
- **filename_ContactSingle.txt** contains all the information for single point contacts.
- **filename_ContactMulti.txt** contains all information for single point contacts.
- **filename_ContactMultiPoints.txt** contains all location information for multipoint contacts.
- **filename_ContactTiff.gat** contains all image files created during export.

Filename represents the full path and name of the output file.

ContactLine

The filename_ContactLine.txt file contains the names of the HIPS track lines and the corresponding index number. Each line in the file is a comma- delimited string terminated by the newline character (\n). Each line has the following format:

Field Name	Field Description	Description
Line Index	4-digit integer	The index number for the line.
Project Name	256 characters (maximum)	The name of the HIPS project that the line belongs to.
Vessel Name	256 characters (maximum)	The name of the HIPS vessel that the line belongs to.
Day Name	256 characters (maximum)	The name of the HIPS day that the line belongs to.
Line Name	256 characters (maximum)	The name of the HIPS line that the contact refers to.

ContactSingle

The filename_ContactSingle.txt file contains the list of single point contact features from the export. Each line in the file is a comma (',')delimited string terminated by a newline character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Contact Index	4-digit integer	The index number for the contact. Index numbers are assigned sequentially from 0 when the export is performed. Guaranteed to be unique for all exported single contacts only.
Key	12 characters	The key for the contact. This is created by concatinating the line name and the contact number together. The first 8 bytes are the first 8 bytes in the line name, and the last 4 bytes are the system generated number assigned to the contact.
Line Index	4 digit integer	The line record number from the corresponding Filename_ContactLine.txt file.
Contact Number	4 digit integer	The system generated contact number assigned to the contact when it was created in the SIPS software (same as the last 4 bytes of the 'key' field).
Creation Date	10 characters	The date the contact was created. Has the mm/dd/yyyy format
Creation Time	10 characters	The time the contact was created. Has the following format: HH:MM:SS.S
Created User ID	3 characters	The user ID entered in the SIPS software at the time of contact creation (usually user initials)
Modified Date	10 characters	The last date the contact was modified. It may be the same as the Creation Date if the contact has not been modified since creation. It has the following format: mm/dd/yyyy
Modified Time	10 characters	The time of the last modification done to the contact. The time may be the same as the Creation Time if the contact has not been modified since it was created. It has the following format: HH:MM:SS.S
Modified User ID	3 characters	The user ID of the user that last modified the contact. The ID may be the same as the Created User ID if the contact has not be modified since it was created.
Contact Prefix	8 characters	The prefix assigned by the SIPS software when the user created the contact. This is normally used to generate the key for the contact.
Feature Code	12 characters	The CARIS Feature code used to represent the point contact.
Contact Type	1 character	This identifies the contact record as either a single point or multi point contact. Single point contacts are represented by a '1' and multipoint contacts are represented by a '2'.
Profile Number	4 digit integer	The profile number that the contact was picked from in the SIPS software.
Across Distance	6.2 Floating point	The across track distance recorded by the SIPS software that the point contact was picked from.
Latitude	13.7 Floating point	The latitude of the point contact.
Longitude	13.7 Floating point	The longitude of the point contact.

Field Name	Field Description	Description
Target Height	6.2 Floating point	The height of the target (in metres).
Target Width	6.2 Floating point	The width of the target (in metres).
Target Length	6.2 Floating point	The length of the target (in metres).
Image Height	4 digit integer	The height of the contact image as defined by the user in the SIPS software.
Image Width	4 digit integer	The width of the contact image as defined by the user in the SIPS software.
Status	1 digit integer	The status of the contact. 0 if the contact is Accepted, 1 if the contact is Rejected.
Remarks	85 (maximum) characters	Remarks entered by the hydrographer about the created contact.

ContactMulti

The filename_ContactMulti.txt file contains the list of multipoint contact features from the export. Each line in the file is a comma (',') delimited string terminated by a newline character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Contact Index	4-digit integer	The index number for the contact. Index numbers are assigned sequentially from 0 when the export is performed. It is guaranteed to be unique for all exported multi point contacts only.
Key	12 characters	The key for the contact. It is created by concatenating the line name and the contact number together. The first 8 bytes are the first 8 bytes in the line name, and the last 4 bytes are the system generated number assigned to the contact.
Line Index	4 digit integer	The line record number from the corresponding Filename_ContactLine.txt file.
Contact Number	4 digit integer	The system generated contact number assigned to the contact when it was created in the SIPS software (same as the last 4 bytes of the ‘key’ field).
Creation Date	10 characters	The date the contact was created. It has the mm/dd/yyyy format
Creation Time	10 characters	The time the contact was created. It has the following format: HH:MM:SS.S
Created User ID	3 characters	The user ID entered in the SIPS software at the time of contact creation (usually user initials)
Modified Date	10 characters	The last date the contact was modified. The data maybe the same as the Creation Date if the contact has not been modified since creation. It has the following format: mm/dd/yyyy
Modified Time	10 characters	The time of the last modification done to the contact. The time maybe the same as the Creation Time if the contact has not been modified since it was created. It has the following format: HH:MM:SS.S
Modified User ID	3 characters	The user ID of the user that last modified the contact. The ID maybe the same as the Created User ID if the contact has not be modified since it was created.
Contact Prefix	8 characters	The prefix assigned by the SIPS software when the user created the contact. This is normally used to generate the key for the contact.
Feature Code	12 characters	The CARIS Feature code used to represent the point contact.
Contact Type	1 character	This identifies the contact record as either a single point or multi point contact. Single point contacts are represented by a ‘1’ and multipoint contacts are represented by a ‘2’.
Target Height	6.2 Floating point	The height of the target (in metres).
Target Width	6.2 Floating point	The width of the target (in metres).
Target Length	6.2 Floating point	The length of the target (in metres).

Field Name	Field Description	Description
Image Height	4 digit integer	The height of the contact image as defined by the user in the SIPS software.
Image Width	4 digit integer	The width of the contact image as defined by the user in the SIPS software.
Status	1 digit integer	The status of the contact—0 if the contact is Accepted, 1 if the contact is Rejected.
Remarks	85 (maximum) characters	Remarks entered by the hydrographer about the created contact.

ContactMultiPoints

The filename_ContactMultiPoints file contains the list of points used to construct the geometry of the multi point contacts listed in the Filename_ContactMulti.txt from the export. Each line in the file is a comma (',')delimited string terminated by a newline character ('\n'). Each line in the file has the following format:

Field Name	Field Description	Description
Record number	4-digit integer	The record number. It is unique throughout the file.
Contact Record Number	4-digit integer	The contact record number that this point belongs to. This corresponds to the Contact Index in the corresponding Filename_ContactMulti.txt file.
Profile Number	4-digit integer	The profile number that the point was computed from in the SIPS software.
Across Distance	6.2 floating point	The across-track distance recorded by the SIPS software that the point contact was picked from.
Accumulated Distance	6.2 floating point	The accumulated distance. This is the distance from the first point in the set, to this point.
Latitude	13.7 floating point	The latitude of the point.
Longitude	13.7 floating point	The longitude of the point.

Glossary

Altitude	The height of the side scan sonar transducer above the sea floor. The transducer is typically mounted on a towed body or the surface vessel, but can also be mounted on an ROV or AUV.
Angle-Varying Gain	Angle-Varying Gain (AVG) correction removes the angular response of sediment from the imagery, normalizing the mean angular intensities, ping-by-ping, with a moving average filter.
Attitude Data	Commonly used to refer to the orientation of the vessel in three dimensions as represented by the Gyro, Heave, Pitch, and Roll sensors. The Attitude Editor displays this data as well as other sensors that consists of simple “time vs. sensor value” data.
Background Data	Georeferenced vector and raster data providing visual context to the bathymetry and side scan data being processed. Examples include CARIS files, BSB raster charts, GeoTIFF orthophoto images, S-57 electronic chart data.
Backscatter	Intensity of sounding returns collected by swath multibeam sounding systems. Backscatter data creates an image of the sea floor that can indicate the bottom type.
BAG	The Bathymetric Attributed Grid (BAG) is an open-source binary file format for transferring gridded bathymetry and uncertainty values between software applications and agencies. The file contains four sections: metadata (in XML format), elevation, uncertainty, tracking, and certification. For more information on BAG, go to http://www.opennavsurf.org/.
BASE Surface	The Bathymetry Associated with Statistical Error (BASE) Surface. A georeferenced TIFF representation of the seabed derived from processed bathymetry and computed uncertainty (error) values. The algorithm applies soundings to the grid based on beam footprint size. A customizable weighting system allows more emphasis in the mean calculation to be placed on the inner nadir beams or on soundings with lower uncertainty values.
Beam	A single depth measurement is produced from a beam. Many beams make up a profile or a ping.

Beam Pattern Correction	Beam pattern correction uniformly removes along-track banding inherent to the sonar beam. This effect is produced by each transducer uniquely. Beam pattern correction relies on a user-generated beam pattern file to identify and remove this effect.
CARIS File	The format for recording vector map data features such as lines, symbols, text, soundings, etc. The map actually consists of set of files.
CATZOC	The zone of confidence (CATZOC) attribute. The CATZOC attribute is part of the S-57 Quality of Data (M_QUAL) object class. It indicates that data meets the minimum criteria for seafloor coverage, and position and depth accuracy as defined by the attribute classification structure. There are five CATZOC classifications: A1, A2, B, C, and D.
Contacts	Point and line features identified in side scan sonar data. Attributes describing the location, size, and description or the target are recorded. Image snapshots are also generated.
CUBE Surface	A Combined Uncertainty and Bathymetry Estimator (CUBE) Surface contains multiple hypotheses representing potential depth variances on the seafloor. As soundings are propagated to nodes—based on distance and uncertainty—a hypothesis is developed. If the there is a significant variation of depth at a node, a new hypotheses is created. A node can contain more than one hypothesis.
Day	One of the days for which there is survey data. Expressed as a Julian day.
Delta Draft	A “time vs. delta-draft” data structure that can be loaded into HIPS and used to model the dynamic squat and lift of the vessel. Also used to store the recorded depth of towfish and ROV/AUV mounted multibeam sonars.
Despeckle	Despeckling uses a calculated value based on neighbouring intensity levels to replace the current pixel’s intensity if it is outside the specified range. The value calculated from the neighbouring intensities can be derived by one of two methods mean or median.
Disambiguation	The process in CUBE of selecting one hypothesis over other hypotheses to represent the surface. Hypotheses can be selected or "nominated" by density, locale, locale and density, or by the nearest value to a previous CUBE Surface.
DpTPE	Depth Total Propagated Error. The difference between the observed or computed depth value of a sounding and its true depth value (at a 95% confidence level).

Edit Layer	A HOB layer created in CARIS Notebook to represent real-world alterations in the survey area.
Field Sheet	A collection of data products, within a defined geographic area, derived from the processed bathymetry and side scan data. A common coordinate system reference is used. Some example products include BASE surfaces, mosaics, tiles, selected soundings, contours, and plot layouts.
Filtering	The process of detecting outliers in the data and setting the status flags to Rejected. Data points are not removed from the HIPS/SIPS format.
Gain	A measure of increase in amplitude of a signal.
GPS Tide	Instantaneous water level height above the sounding datum derived from the observed GPS ellipsoid heights of the navigation antenna. Reduced for sensor offsets, vessel motion, known sounding datum height above the ellipsoid, and other parameters.
Generic Data Parser	A program for converting recorded ASCII data sources into the HIPS/SIPS format. New survey lines can be generated for single beam bathymetry, or the program can update a sensor within an existing HIPS or SIPS project.
HOB	The Hydrographic Object Binary (HOB) file is a spatially referenced dataset that is used to support the internal structure of S-57 data in CARIS products. The HOB file does not require a CARIS vector map and contains the point, line, and area geometry for hydrographic objects.
HzTPE	Horizontal Total Propagated Error. The difference between the observed or computed position value of a sounding and its true position value (at a 95% confidence level).
HVF	HIPS Vessel File. A vessel configuration file in XML format that consists of a list of sensors with their physical and calibration measured offsets, plus any error values. These are applied to the observed data during processing. The HVF supersedes the Vessel Configuration File (VCF).
Hypothesis	A representation of depth in a BASE Surface. The Hypotheses that are displayed in the Surface are selected or "nominated" through disambiguation. Hypotheses that were not selected remain as alternative hypotheses and can be nominated in the Subset Editor.
Layback	If a towed sensor is used during a survey, the position of that sensor can be calculated as a horizontal “layback” from the position of the towing vehicle. The sensor layback is computed from the tow wire length and sensor depth.

LADS	Laser Airborne Depth Sounder (LADS) is a lidar system developed by Tenix Corporation.
Lidar	Light direction and ranging uses laser technology to calculate bathymetry.
Line	A single pass of the survey vessel over the survey area. Time referenced sensor data is continuously recorded.
Marker Layer	A HOB layer created in CARIS Notebook for graphically identifying areas of interest and attaching notes, text and other information. The Marker layer is created as a sub-layer of an Edit layer.
Merge	The process of calculating final positions and depths for soundings, based on all relevant inputs such as observed depths, navigation information, vessel dynamics such as gyro, heave, pitch and roll, and tide.
Mosaic	A georeferenced raster image created from one or more side scan survey lines.
Outliers	Unwanted data resulting, for example, from incorrect sea floor detection.
Ping	A single output pulse of a sonar system
Profile / Swath	A set of soundings, approximately perpendicular to the ship's track, which is produced by the swath or sweep sonar.
Product Surface	A shoal-biased bin generated from a BASE Surface. Data is down-sampled so that finer details of the seafloor are not visible, but the shoals are still maintained. Product surfaces can be used to create navigation products, such as Electronic Nautical Charts.
Project	A survey area with data collected by one or more survey vessels over one or more days.
Publication Designer	The Publication Designer is an application developed by CARIS for formatting templates. You can use this application to insert dictionary tags, and add text and graphics. The application is opened directly from the Start button on the Windows taskbar or from a shortcut on the desktop.
Raw Side Scan	Time indexed imagery profiles where the across-track scale is time in milliseconds.
S-44	S-44 is the international standard developed and maintained by the International Hydrographic Organization (IHO) that sets the minimum standard for hydrographic surveys. The standard classifies surveys into survey orders (Special Order, Orders 1, 2, 3) based on an area's importance for safety of surface navigation.

S-57	S-57 is the international standard developed and maintained by the International Maritime Organization (IMO). It is used by hydrographic data producers to describe how real-world entities are stored digitally for transfer among different computer platforms between the various hydrographic offices.
Session	A record of all data that is currently open in the primary application. Typically contains a list of survey line names, background raster images, and field sheets that are currently open. Used to re-open an integrated dataset from a previous working session.
SHOALS	Scanning Hydrographic Airborne Lidar Survey (SHOALS) is a lidar system owned and operated by the Joint Airborne Lidar Bathymetry Center of Expertise.
Slant Range	Raw travel time used to compute sounding depth (accompanied by receive angle).
Slant Range Corrected Side Scan	Time-indexed imagery profiles generated from raw side scan data using the measured altitude of the transducer. The across track scale is distance in metres.
Sounding	A measured depth of water.
Sound Velocity Correction	The process of applying rigorous refraction corrections to the raw travel time/angle bathymetry data using recorded sound velocity profiles.
Status Flag	The indicator of the acceptance or rejection of a data point during cleaning.
Subset	An area which encloses some or all of a survey project boundaries. It is used during sounding cleaning to limit the number and extent of soundings being loaded into memory.
Surface Cleaning	A statistical process of detecting bathymetric outliers within an area. A least square polynomial regression algorithm is used.
Templates	Templates are forms that display session data. Session data is added using object tags from in the HIPS dictionary. When the template is opened in HIPS and SIPS, the object tags are filled with data from the open session. Other features, including text and graphics, can also be added. Templates can be built using the Publication Designer.
Template Editor	The Template Editor is a CARIS application for creating and modifying borders, geographic grids, and scale bars. The Template Editor is used in conjunction with the Publication Designer for creating and editing templates. It is opened as a separate application from the Start menu on the Windows taskbar, or from a shortcut on the desktop.

Tiles	A representation of the sea floor consisting of interlocking tiles of varying sizes. Tile positions are defined by a Morton Number scheme. Tiles can carry a variety of attributes generated from the data within the area of the tile.
Time-Varying Gain	Time-Varying Gain (TVG) adjusts the signal intensity by applying a non-uniform, time-dependent gain. TVG is used to compensate for the attenuation that occurs due to absorption and spreading as the sonar beam travels back to the receiver. Since these returns are received over a predictable and constant time period, a time-varying curve can used to increase gain to compensate for the decayed sonar return.
TrueHeave	A heave filtering function developed by Applanix. TrueHeave employs a bi-directional heave filter to remove artifacts from heave data. Values are computed from real-time data and have to be imported separately to the track line.
TPE	Total Propagated Error. TPE is derived from a combination of all individual error sources and is used to calculate horizontal and vertical uncertainties for soundings.
Vessel	A survey platform, whether it be a ship, a towfish, an ROV/ AUV, or an aircraft.
Weighted Grid	A georeferenced TIFF representation of the mean sea floor depths derived from processed bathymetry. The algorithm applies soundings to the grid based on beam footprint size. A customizable weighting system allows more emphasis in the mean calculation to be placed on the inner nadir beams. The weighted grid was superseded by the BASE Surface.

Index

Symbols

*.tid, 213
*.zdf, 213
.bsd, 583
.grd, 583
.ssd, 583

Numerics

16-Bit Shift Facto, in mosaic, 547
16-Bit Shift Factor, 499
16-bit side scan display, 499
2-D Display icon, 62
2-D view in Subset Editor, 393
3-D display
    camer informattion panel., 61
    camera information panel, 53
    colour map panel, 53
    control axes, 58
    fly through controls, 60
    navigate, 60
    navigate with 2-D icon, 62
    overview, 48
    Subset Editor, 389
    Swath Editor, 312–316
    Vertical exaggeration, 59
    vertical exaggeration properties, 55
    Vessel Editor, 119
3-D display fly-through options, 63
3-D Display Properties, 53
3-D fly-through
    playback, 64
    record, 63

A

Across-track display, 497
add line contact, 540
add point contact, 540
angle-varying gain, 534, 560
ASCII file, export data to, 605
Atlas format, conversion options for, 179
attitude data, 272
    accept data, 282
    difference line, 280
    display options for editor, 87
    draw bars, 276
    filter, 217
    filtering, 284–285
    interface, 272
    invert view, 276
    load Applanix files, 221
    moving average line, 280
    open editor, 272
    query line, 283
    reject-break interpolation, 281
    reject-with interpolation, 281
    reset rejected data, 247
    scale data, 276
    select attitude sensors, 274
    smoothing, 278–280
    threshold lines, 280
    time, 275
    time frames, 274
    TrueHeave, 219
    vertical view, 276
    view rejected soundings, 207
auto cursor mode, 321
auto save, 76
Automatic BASE Surface Update, 76
automatic depth filtering, 232
automatic recentre, 310
Average compression, 497
AVG correct, 534

B

Background data
    open, 31
backscatter, update, 242
barcheck function in Single Beam Editor, 335
BASE surface
    add survey data, 423
    attribute properties, 414
    close, 422
    combine surfaces, 429
    create, 408
    custom layer, 431
    delete, 427
    finalized surface, 427
    holiday layer, 432
    Interpolate, 425
    open, 422
    overview, 402
    properties, 413
    quality control, 255
    quality report, 432
    range filters, 415
    range weight, 404
    recompute, 423

remove survey lines, 424
swath angle weight, 406
uncertainy weight, 405
update units, 421
view, 412
weighted grids, comparison with, 407
batch processing
new file, 568
open editor, 566
open HBP, 568
overview, 566
save file, 568
select process, 569
tasks, 571
Bathymetric Attributed Grid, export data to, 593
bathymetry,use data in side scan, 548
beam pattern correction, 529
Benthos C3D data, 197
bounding box, subset, 387

C

calibration
example procedures, 488
overview, 482
perform, 486
subsets, create in HDCS, 484
workflow, 483
camera information panel, for 3-D display, 53
Camera Information Panel, using, 61
CARIS map
export soundings to, 608
GSF data, 600
open, 31
symbolize, 76
CHS (Canadian Hydrographic Service), 3
classify lines, 206
Close
Field Sheets, 352
CMAX format, conversion options for, 180
Coda format, conversion options for, 180
Colour Map Editor, 417
Colour Map Panel, 3-D display, 53
Colour Mapping, side scan data, 507
Colour Range Editor, 419
Colour Table Editor, 371
command access from toolbars, 24
Compression
side scan display, 497
Skip, 497
Compression, image
Average, 497
maximum, 497
configuration file
new, 632
open, 631
save, 632
save as, 632
configuration report, 9, 9
contacts
add line contact, 540
add point, 540
change values, 542
create, 539–544
export, 597–599
move, 542
recompute position, 544
select, 542
Contacts, delete, 543
Contacts, view while scrolling, 502
control axes, 58
Control window
overview, 12
select track lines, 46
show or hide layers, 40
track lines, closing, 72
Controller Panel properties,for 3-D display, 54
conversion
Atlas, 179
CMAX, 180
Coda, 180
EdgeTech, 180
Elac, 181
Furuno, 182
Generic Data Parser, 628
GeoAcoustics, 183
GSF, 185
Hypack, 186
Imagenex, 187
LADS, 188
LAS, 188
MarineSonics, 189
Navitronics, 189
Qmips, 189
Reson, 189
Scripps, 189
SDF, 190
Seabeam, 190
Seafalcon, 190
SEGY, 191
SHOALS, 191
Simrad, 193
SWATHPlus, 194
UNB (Reson), 194
Winfrog, 194
XTF, 196
Conversion Wizard, 170–177
coordinates, on status bar, 23
Copy command, 67
Coverage outline, side scan, 496
Create Graph, 18
CUBE
create, 441
hypothesis editing, 449
nominate hypotheses, 452
overview, 438

properties in Subset Editor, 449
reject hypotheses, 454
cut command, 66

D

data conversion, 170
decimal point precision, 23
Delete
new toolbar, 27
delete contact, 543
delta draft, loading, 249
designated sounding flag, 318, 399
designated soundings, 208
Despeckle, 561
despeckle side scan display, 538
detrend option, 306
development partners, 3
digitizing sonar altitude, 521
directories, set location for, 97
directory structure for projects, 109–112
Display Critical Soundings, 210
Display options
image files, 32
Display Properties, Profile, 363
Display tide zones, 213
display tide zones, 214
Display window
geographic grid, 80
measure distance, 69
overview, 11
overview display, 40
pan, 39
projected grid, 84
refresh display, 40
screen captures, 47
set background colour, 79
show distance when selecting, 77
show line when selecting, 77
show or hide layers, 40
side scan outline, 79
zoom, 37
distance, measuring in Display window, 69
dynamic draft data in Vessel Editor, 140

E

EdgeTech format, conversion options for, 180
Elac format, conversion options for, 181
Enable during playback, 79
Enable range display in Side Scan editor, 498
environment options
S-57, 101
event marks, 328
export HIPS data, 586
Export Wizard, 586

F

Field Sheet Editor, 354
field sheets
close, 352
new, 349–351
open, 352
overview, 348
properties, viewing, 353
set as active, 352
sounding display size, 75
filtering options, mosaic, 551
filters
attitude data, 284–285
single beam, 243–246
swath, 230–232
total propagated error, 236
filters, mosaic, 551
finalized surface, 427
find soundings, 308
fixed column, 629
flag
designated soundings, 208
Fly-through, record, 63
font
set for measure distance display, 79
free form, 629

G

Generic Data Parser, 628–632
GeoAcoustics format, conversion options for, 183
geographic grid, 80
GeoTiff
capture screen as, 47
GPS (Global Positioning System) Tide, 250
Graph Controls, 19
Graph Tab, 18
Graph,create, 18
Graph,display options, 19
graphs, 18
grid
geographic, 80
projected, 84
GSF (Generic Sensor Format)
conversion options for, 185
export HIPS data to, 612
export to CARIS map, 600
export to HOB, 600
gyro sensor data in Vessel Editor, 135

H

HDCS (Hydrographic Data Cleaning System) create subsets in, 484
heave sensor data in Vessel Editor, 137

HOB (Hydrographic Object Binary) files
export GSF to, 600
export soundings to, 613
object filters, 34
open, 34
holiday layer for BASE Surface, 432
HTF (Hydrographic Transfer Format), export layer data to, 615
HVF (HIPS Vessel File) *see Vessel Editor*
Hypack format, conversion options for, 186

I

I/O library, 6
identifier, 630
image classification, 555
Imagenex, conversion options for, 187
images
open background, 31
properties, 32
screen captures, 47
Interpolation
side scan data, 525
interpolation
set for navigation data, 229

K

keyboard shortcuts
shortcuts, 70

L

LADS format, conversion options for, 188
LAS data format, conversion options for, 188
lasso
use in Display window, 44
lasso, select by, 44
layers, 40
licensing options, 5
lidar
accept data, 479
detected and alternative data, 477
display options, 83
find soundings, 476
LADS conversion options, 188
open data, 475
overview, 470
quality flags, 470
query data, 477
reject data, 479
Shoals conversion options, 191, 191
status flags, 470
toolbar, 473
waveform views, 473
Load
Delta Draft, 249
Load Attitude/Navigation Data, 221
Load Tide, 215
Load TrueHeave, 219
lock subset area, 388

M

Manual digitizing,side scan data, 524
MarineSonics data format, conversion options for, 189
Maximum compression, 497
Measure Distance, 543
Measure Shadow, 543
merge
lines, 253
merge files, 253
middle mouse select, 321
mosaic
16-bit shift factor, 547
add lines to, 554
change outline colour, 553
determine data source, 549
direction of shadow, 550
extrapolate, 549
image classification, 555
Interpolation, 549
new, 546
orientation source, 547
remosaic, 555
remove lines from, 554
set resolution and colour, 547
toggle name, 553
use bathymetry to register side scan imagery, 548
mosaic filtering options, 551
mosaics
create, 545–552
export, 619
intensity properties, 553
properties, 552
mouse information, 28
Move toolbars, 25
multiplier, 630

N

navigate the 3-D window, 60
navigation
accept data, 269
bezier interpolation, 265
connect points, 264
editor interface, 262
linear interpolation, 265
load Applanix files, 221
navigation antenna data in Vessel Editor, 135
point display, 263

point symbol selection, 76
query data, 270
reject-break interpolation, 269
reject-with interpolation, 269
reset rejected data, 247
resize points, 263
scaling, 263
set interpolation, 229
speed jumps, 267
time jumps, 268
view rejected soundings, 207
Navitronics data format, conversion options for, 189
new projects, 150–155
New Publication, open template, 576
NOAA (National Oceanic and Atmospheric Administration), 3
NOS final tide format, 652
NOS preliminary tide file format, 651

O

Ocean Mapping Group, 3
offset, 630
Open
Background Data, 31
file types, 31
Open Project, 29
Options - General, 75
outline colour
mosaic, 553
Output Tab, 15

P

pan
3-D display, 60
commands, 39
forward, 38
previous, 38
set pan factor, 75
Swath and Subset Editors, 316
using coordinates, 39
Vessel Editor, 120
Paste command, 68
pick aperture, 75
ping
select by number, 504
select range, 504
select single, 504
Ping selection,from start and end numbers, 499
pitch data in Vessel Editor, 138
Playback 3-D fly-through, 64
Playback in Side Scan Editor, 501
Playback, view contacts during, 502
Playback,move side scan outline during, 79
processed mode, 496
Profile
Display Properties, 363
Profile Tab, 21
profiles
axis settings, 365
colour, 364
create, 362
display, 370
export, 368
horizontal exaggeration, 363
legends, 367
overview, 362
pan, 367
rotate, 367
vertical exaggeration, 363
zoom, 367
project
open, 29
projected grid, 84
projects
auto save, 76
auto update for UTM, 76
batch processing, 566
close, 72
convert raw data, 170
directories, 109–112
export data, 591–618
merge files, 253
new, 150–155
properties, 157
rename folders, 156
save, 72
session files, 159
properties
BASE Surface, 413
field sheets, 353
images, 32
mosaics, 552
tiles, 357
track lines, 30
Properties window, 14
pseudo timestamp, 630
Publication Designer, 574
Publication Objects, 576
publication template, 574–580
Publication Viewer, 580
commands, 580
Publish, 577
purge rejected, 543

Q

QC report, 255
QC Reports tab, 21
Qmips format, conversion options for, 189
QPS, 197
quality control, 255

quality report for BASE surface, 432
Query
    S-57 data, 35
query line, 270
Query line data, 205
Query tab, 16
query tiles, 361

R

Range display, 498
range weight, 404
range-select side scan data, 504
raw mode, 496
Reapply, 528
Recompute BASE surface, 423
Recompute mosaic, 555
record 3-D fly-through, 63
recycle bin, 161
reference position for sensor equipment, 115
Refraction Editor, 322
refresh
    automatic, 76
Refresh command, 40
rejected contact, 543
rejected soundings, view, 207
remosaic, 555
rename folders, 156
Reposition window, 22
resize
    subset, 387
Resize window, 22
Reson format, conversion options for, 189
restart cleaning, 247
Revert, 528
roll data in Vessel Editor, 139
root path for directories, 98

S

S-44
    quality control, 255
    set colour display in Subset Editor, 82
    TPE filtering, 236
S-57 data
    display in Display window, 34
    environment settings, 101
    object filters, 34
    query data, 35
    S-52 display options, 103
    set CATZOC colour in Subset Editor, 82
scale bar in Display window, 86
screen captures, 47
Scripps format, conversion options for, 189
SDF format, conversion options for, 190
Seabeam format, conversion options for, 182, 190
Seafalcon format, conversion options for, 190
SEGY format, conversion options for, 191
select side scan data, 504
selected soundings, 371
selection tools
    Control wndow, 46
    current display, 42
    deselect, 46
    lasso select, 44
    next line, 45
    previous line, 45
    range, 42
    select all, 42
    Side Scan Editor, 504
session files, 159
SHOALS format, conversion options for, 191
side scan
    angle-varying gain, 560
    digitize data manually, 524
    interpolate data, 525
    mosaics, 545–552
    signal display, 515
    towfish track line colour, 79
side scan data processing workflow, 492
Side Scan Editor
    , 494–544
    16-bit display, 499
    accept data, 511
    across track display, 497
    add to mosaic, 554
    angle-varying gain, 534
    automatic scrolling, 501
    beam pattern correction, 529
    colour mapping, 507
    colour options, 507
    compression values, 497
    Compression, Skip, 497
    contacts, 539–544
    coverage outline, 496
    delete contact, 543
    despeckle, 538
    digitizing, 521
    display modes, 496
    fish altitude, 521
    measure distance, 509
    measure distance or shadow, 543
    measure shadow, 510
    overview, 492
    playback, 501
    processed mode, 496
    processing, 494
    purge rejected contact, 543
    query data, 511
    range display, 498
    raw mode, 496
    reapply, 528
    recompute contact position, 544
    recompute mosaic, 555
    recompute towfish navigation, 519

reject data, 510
remove from mosaic, 554
revert, 528
select a single ping, 504
select contacts, 542
selection, 504
signal display, 515
SIPS template wizard, 557–564
slant range correction, 525
specify start and end of ping selection, 499
time-varying gain, 535
toolbars, 500
view contacts during playback, 502
window, 496
workflow, 492
zoom, 503
Side Scan Editor display
16-bit shift factor, 499
Simrad format, conversion options for, 193
single beam data
accept data, 333
add depths, 331
auto cursor mode in editor, 334
automatic depth filtering, 243–246
barcheck function, 335
beam-to-beam slopes filter, 246
change depths, 330
change sounding position from the Worksheet window, 334
change sounding status from Worksheet window, 333
depth filter, 246
event marks, 328
horizontal labels for, 328
horizontal scaling, 327
moving average filter, 245
primary and secondary data, 329
query data, 333
reject data, 332
restart cleaning, 247
toolbar, 327
vertical scaling, 327
view rejected soundings, 207
single beam data in the Vessel Editor, 141
single beam filtering, 243–??
SIPS template wizard, 557–564
Skip, 497
Slant Range Correction, 559
slant range correction, 525
smoothing sensor data, 278–280
Sonar altitude,digitizing, 521
sound velocity correction
see also SVP Editor
nearest in distance option, 226
nearest in time option, 226
previous in time option, 226
sound velocity processing
overview, 223
soundings
colour code in Subset editor, 386
designated., 208
display critical, 210
set status in Subset Editor, 398, 398
status bar
cursor coordinates, 23
decimal point precision, 23
overview, 23
precision, 95
subset
drawing bounding box, 387
resize, 387
Subset Editor
2D View, 393
2D view perspective, changing, 394
3D View, 389
close, 396
data status, ??–379
designated soundings, 399
examined status flag, 400
lock subset area, 388
open, 380
outstanding status flag, 400
overview, 374
query, 400
reject data, 398
resize subset, 387
set sounding status, 398, 398
sounding colour options, 386
subset tiles, 376–??
toolbar, 384
vertical exaggeration in 2D View, 394
view information, 384
workflow, 375
Subset Editorsubset tiles, ??–379
subsets
accept data, 399
CATZOC colour settings, 82
CUBE Surfaces, 449
reset rejected data, 247
S-44 survey order colour setting, 82
view rejected soundings, 207
Sun Position, 550
surface cleaning
histogram, 466
overview, 459
parameters, setting, 461–463
query tile soundings, 465
query tile surface, 464
tiling, overview, 458
SVP (Sound Velocity Profile), 223
apply correction, 225
barcheck function, 335
location symbol selection, 76
SVP Editor
depth scale, 289
edit existing data, 293
exit, 296
interface options, 291

new files, 290
new profiles, 295
remove profile, 294
SVP files, 224
SVP locations, display, 228
SVP pole offsets in the Vessel Editor, 142
swath angle weight, 406
swath cleaning, 230–??
Swath Editor
3D View, 301
accept data, 318
Amplitude View, 302
auto cursor mode, 321
beam and swath location, 308
beam display size, 306
change sounding status from Worksheet window, 320
close, 300
depth level labels in Side View and Rear Views, 304
detrend option, 306
examine data, 307
find soundings, 308
middle mouse select, 321
open, 299
Plan View, 301
port and starboard beams, displaying, 303
Profile View, 302
query data, 320
Rear View, 302
Refraction Editor, 322
reject data, 317
reject swaths, 318
reset rejected data, 247
Side View, 302
swath coverage outline, 310
swath outline, 79
toolbars, 300
view rejected soundings, 207
swath transducer location in Vessel Editor, 134
SWATHPlus format, conversion options for, 194
sweep transducer location in Vessel Editor, 143
symbology,track line, 30

T

Template
New Publication, 576
Template Editor, 574, 583
Terrain Axes properties, 56
terrain properties, 55
text files (Generic Data Parser), 633
tide
see also Tide Editor
display tide zones, 214
GPS tide, 250
load tide, 215
tide zone files, 653
weighted average option, 214
Tide Editor
create new tide files, 342
edit table data, 340
enter new data, 340
exit, 345
open, 338
open file, 338
scale, 339
search time gaps, 340
show or hide points, 343
time interval, 339
tool tips, 343
tide file formats
NOS final, 652
NOS preliminary, 651
Tide Files, 213
Tide files, 213
Tide stations, 214
Tide Zone files, 213
tiles
see also surface cleaning
close, 356
display options for the View window, 357
open, 356
overview, 355
properties, 357
query display, 359
query tiles, 361
Time Correction, 116
time error in Vessel Editor, 116
time stamp in Vessel Editor, 116
time-varying gain, 535
tool tip display, 76
toolbars, 24
about, 24
buttons, modifying appearance of, 25
custom, 26
move, 25
removing from desktop, 25
Side Scan Editor, 500
Single Beam Editor, 327
Subset Editor, 384
Swath Editor, 300
types, 24
undock and float, 25
towed sensors location data in Vessel Editor, 145
towfish navigation, recompute, 519
towfish trackline colour, 79
TPE (Total Propagated Error)
compute, 234
filter, 236
vessel editor values, 147
TPE filtering
S-44, 236
track lines
classify lines, 206
close single lines, 72

colour, 31, 79
colour banding, 79
Control window, select from, 46
current display, select by, 42
deselect, 46
lasso select, 44
opening a single line, 29
properties, 30
query line, 205
range, select by, 42
select all, 42
select next line, 45
select previous line, 45
symbology, 30
TrueHeave, 219
TVG, 535

U

UNB (Reson) format, conversion options for, 194
UNISIPS, export data to, 618
units, update BASE Surface, 421
units,decimal point precision, 23
University of New Hampshire, 3
Use Measure tools, 543
UTM, auto update, 76

V

Vertical Exaggeration properties, for 3-D display, 55
Vessel Editor
3-D vessel outline, 119
colour options, 123
coordinate system, 114
dynamic draft, 140
ellipsoid, 123
gyro, 135
heave, 137
navigation, 135
new vessel file, 125–131
open file, 118
pitch, 138
reference position, 115
roll, 139
save file, 118
single beam, 141
swath transducer location, 134
sweep transducers location, 143
time error, 116
time stamp, 116
towed sensors location, 145
TPE, 147
vessel outline, 121
view active sensors, 132
view sensor data, 132
waterline data, 146
vessel properties, 157

W

waterfall window, 496
waterline data in Vessel Editor, 146
weighted grids and BASE surfaces, 407
windows
3-D Display, 48
Control, 12
Display, 11
Properties, 14
reposition and resize, 22
Worksheet, 15
Winfrog format, conversion options for, 194
Worksheet window
Graph tab, 18
Output tab, 15
Profile tab, 21
QC reports, 21
Query tab, 16
Worksheet window, overview of, 15

X

XTF format, conversion options for, 196

Z

zerotide, 213
Zone Definition File, 213
zoom
3-D display, 60
constant zoom option, 76
forward, 38
previous, 38
Side Scan Editor, 503
Swath and Subset Editors, 316
using scale, 39
Vessel Editor, 120
zoom factor, 75
zoom in, 37
zoom out, 38
zoom tool, 37